Further development of 2D XFEM in VAST

Lei Jiang and Merv Norwood
Martec Limited

Prepared By:
Martec Limited
400-1800 Brunswick Street
Halifax, Nova Scotia
B3J 3J8 Canada

Contract Project Manager: Lei Jiang, 902-425-5101 ext 228
Contract Number: W7707-125422/001/HAL CU01
CSA: Ian Thompson, 902-426-3100 ext 358

Defence Research and Development Canada

Contract Report
DRDC-RDDC-2014-C88
MARTEC TR-13-30
July 2013

Principal Author

Original signed by Lei Jiang

Lei Jiang

Senior Research Engineer

Approved by

Original signed by Neil Pegg

Neil Pegg

Head/Warship Performance

Approved for release by

Original signed by Calvin Hyatt

Calvin Hyatt

Chair/Document Review Panel

Abstract

This report is concerned with further development and validation of the XFEM fracture mechanics analysis capability in VAST. In the present work, some of the limitations in the previous version of the XFEM fracture element were removed. The improved XFEM capability permits curved shell geometry with arbitrary orientations and cracks that pass through element corners. Pre- and post-processing capabilities for the XFEM element were developed in the TRIDENT system where the conventional and XFEM fracture elements are treated under a unified framework. In order to demonstrate the suitability of the XFEM element for spectral fatigue crack propagation analyses, a case study was performed using a practical problem involving crack growth in a frigate under certain operation conditions. The fatigue analysis was performed using the unit panel method where the stress intensity factors must be provided for each of the unit panel and rigid-body acceleration load cases. The crack increments predicted by the stress intensity factors from XFEM and conventional fracture elements are in good agreement and they both agree well with the measured data.

This page intentionally left blank.

Executive summary

Further Development of 2D XFEM in VAST

L. Jiang and M. Norwood; DRDC-RDDC-2014-C88; DRDC– Atlantic Research Centre; July, 2013.

Introduction: The modeling of fracture and material damage has been a problem of significant interest in solid mechanics for a long time. Many finite element formulations have been proposed for fracture mechanics analyses. However, all the classical finite element approaches require the crack be explicitly modeled in the finite element mesh. This is challenging for problems with complicated geometry and for crack propagation, continuous remeshing is required. In order to minimize the requirement of remeshing during crack propagation analysis, a new finite element formulation, named the extended finite element method (XFEM), has been developed. In this method, the classical finite element approximation is enriched by a discontinuous function and the asymptotic displacement field around crack tips. As a result, cracks are permitted in the interior of elements. The XFEM was implemented in VAST during previous contracts, but it still contained some limitations. For instance, the cracked plate had to be planar and reside in the global X-Y plane. In addition, the crack was not permitted to pass through corners of the XFEM element. The objective of the present contract was to remove these limitations, develop pre- and post-processing capability for XFEM in TRIDENT and demonstrate the suitability of XFEM for practical crack propagation analysis through a case study.

Results: Removal of the limitations in the previous version of the XFEM element was achieved by performing all operations in the element local coordinate system, such as the differentiation of the enrichment field and applying internal constraints along the element normal direction. The allowance of the crack to pass through element corners required adjustment of the nodal values of the enrichment functions. The pre- and post-processing capability for XFEM was implemented in TRIDENT under a unified framework with the conventional fracture element to reach a high level of usability. All these new developments were extensively verified by numerical example. A case study was finally performed using a practical spectral fatigue crack propagation problem where all the steps of analysis were carried out though the TRIDENT GUI. The XFEM results obtained for different sea states were in good agreement with those from the conventional fracture element and the actual measurements.

Significance: The modifications significantly improved the applicability of the XFEM element to practical problems where cracked structural members are often slightly curved and arbitrarily oriented in the 3D space. The development of the pre- and post-processing capabilities for XFEM in TRIDENT provided a useful tool for applying the XFEM element to practical problems. The suitability of the XFEM fracture element for solving practical spectral fatigue crack propagation problems was successfully demonstrated through the case study.

Future plans: The case study indicated that the current XFEM fracture element is unable to treat branching cracks involving intersecting plate members which are often seen in ship structures. Due to this limitation, approximations may have to be made when generating the XFEM model. It is recommended that this limitation be removed in future development of the XFEM capability.

This page intentionally left blank.

Table of contents

Abstract …….. i
Executive summary iii
Table of contents v
List of figures vi
List of tables viii
1 Introduction.. 1
2 Extension of XFEM to Curved Shell Surface With Arbitrary Orientations 2
2.1 Generation of test cases 2
2.2 Modifications to the XFEM fracture element.. 7
2.3 Verification of the modified XFEM element 8
3 Extension of XFEM to Allow Crack to Pass Element Corners 9
4 Implementation and Verification of XFEM Capability in TRIDENT 15
4.1 Pre-processing in TRIDENT 15
4.2 Post-processing in TRIDENT... 21
4.3 Verification of XFEM capability in TRIDENT 23
5 Case Study 27
5.1 Problem definition 27
5.2 The global and local finite element models... 27
5.3 Spectral load analysis 31
5.4 Calculation of stress intensity factors 31
5.5 Crack propagation analysis.. 32
6 Conclusions... 34
References 35

List of figures

Figure 1: Rotated XFEM models for a plate with a 45^o slanted edge crack. 3

Figure 2: Generation of curved XFEM models. 5

Figure 3: Curved XFEM models for a plate with a circular embedded crack. 6

Figure 4: Cases for crack passing one or more corners of a XFEM element. 9

Figure 5: Plate with angled center crack. 11

Figure 6: XFEM models of different levels of refinement for a center cracked plate with a crack length a=1 and an orientation angle β=0^O. 12

Figure 7: XFEM models of different levels of refinement for a center cracked plate with a crack length a=1 and an orientation angle β=30^O. 13

Figure 8: XFEM models of different levels of refinement for a center cracked plate with a crack length a=1 and an orientation angle β=45^O. 14

Figure 9: Creation of fracture elements by converting regular shell elements. 16

Figure 10: Examples for definition of crack lines and tips for regular and XFEM fracture elements. 17

Figure 11: Using the split model operation in TRIDENT. 18

Figure 12: TRIDENT menu for defining fracture element enrichment formulation data. 19

Figure 13: Verification of regular fracture element enrichment data using the inquire feature in TRIDENT. 19

Figure 14: Verification of XFEM fracture element enrichment data using the inquire feature in TRIDENT. 20

Figure 15: Verification of XFEM fracture element enrichment data using the verify feature in TRIDENT. 20

Figure 16: TRIDENT stress component list for the four node fracture element. 21

Figure 17: TRIDENT generated plots from a fracture analysis using regular enriched fracture elements IEC=18. 22

Figure 18: TRIDENT generated plots from a fracture analysis using XFEM enriched fracture elements IEC=68. 22

Figure 19: TRIDENT generated XFEM models for a plate with a 45^O slant edge crack. 24

Figure 20: TRIDENT generated XFEM models for a plate with a 30^O center crack. 24

Figure 21: TRIDENT generated XFEM models for a plate with a circular center crack. 25

Figure 22: TRIDENT generated XFEM models for a plate with a kinked center crack. 25

Figure 23: Global finite element model. 28

Figure 24: Crack Details. 28

Figure 25: Top-down analysis procedure. 29
Figure 26: Local finite element model with crack details. 29
Figure 27: Crack details in local finite element model based on conventional fracture elements... 30
Figure 28: Crack details in local finite element model based on XFEM elements. 30

List of tables

Table 1: Stress intensity factors obtained using the original and rotated XFEM models for the test case involving a 45^{o} slanted edge crack. 4

Table 2: Stress intensity factors obtained using the original and curved XFEM models for the test case involving an embedded circular crack. 5

Table 3: Test results for XFEM models with crack passing through element nodes. 10

Table 4: Comparison of stress intensity factors obtained using different XFEM models for selected test cases. 26

Table 5: Comparison of stress intensity factors obtained using different XFEM models for test case with a kinked centre crack. 26

Table 6: Sea state cases considered. 31

Table 7: Sea states definitions by significant wave height and peak modal period – LR 2005 31

Table 8: Comparison of stress intensity factors (MPa√mm) obtained using conventional and XFEM fracture elements for the first twenty-five unit pressure load cases. 32

Table 9: Observed and predicted crack growth over 2.78 hours under different sea states. 33

1 Introduction

The modeling of fracture and material damage has been a problem of significant interest in solid mechanics for a long time. This is because crack initiation and propagation are important factors that need to be considered in design and maintenance of practical engineering systems. One example is the accurate prediction of fatigue crack propagation in ship structures subjected to cyclic loading. Many finite element formulations have been proposed for fracture mechanics analyses over the years. However, all the classical finite element approaches have a common disadvantage. They require the crack to be explicitly modeled in the finite element mesh, which can be very challenging for complex engineering structures with curved crack geometry. In addition, to simulate crack propagation, continuous remeshing has to be performed and repeated mapping of the field variables, such as stresses and strains, are required between the old and new meshes which may raise concerns on accuracy of the numerical solutions.

In order to minimize the requirement of remeshing during crack propagation analysis, a new finite element formulation, named the extended finite element method (XFEM), has been developed. In this method, the standard displacement field in the finite element method is enriched by applying a discontinuous displacement function along the crack line and an asymptotic displacement field around the crack tips based on a recently developed mathematical formulation named partition of unity. Up to the present time, XFEM has been applied to a wide variety of fracture mechanics problems, including arbitrary branching and interaction of multiple cracks, dynamic and fatigue crack propagation and crack evolution in shell structures undergoing large displacements and rotations. In addition, XFEM has been extended to non-planar 3D crack growth simulations.

Comparing with earlier numerical methods for fracture mechanics, XFEM has a number of advantages, including (a) it does not require the cracks be explicitly modeled, so no remeshing or minimal remeshing is needed for crack propagation; (b) it is a finite element method, so it can be implemented in existing general-purpose finite element programs, such as VAST; (c) in contrast to boundary elements, it is readily applicable to non-linear problems; and (d) in contrast to finite elements with remeshing, it does not require as many projections between different meshes.

In two previous contracts from DRDC Atlantic, the extended finite element method (XFEM) was implemented in VAST for solving two-dimensional fracture mechanics problems. This earlier implementation utilized linear elastic fracture mechanics solutions as the crack tip enrichment functions. Consequently stress intensity factors were introduced as nodal variables of the enriched elements and solved directly along with nodal displacements. The XFEM capability in VAST was later extended to permit curved and kinked crack geometry. In addition, the domain version of the interaction integral, a special variation of the J-integral suitable for mixed mode problems, was also implemented to provide a means to assess the accuracy of the direct approach.

The main objective of the present contract was to further improve the usability of the previously developed XFEM analysis capability in VAST for practical engineering analyses. These include modifications of the XFEM fracture element to permit arbitrary orientations, curvatures in the cracked plates and more general situations in crack-mesh interaction. In addition, the TRIDENT program was enhanced to provide pre- and post-processing capabilities for the XFEM element. Finally, a case study was performed using a practical spectral fatigue crack propagation problem to demonstrate applications of the XFEM capability. All these studies are described in this report.

2 Extension of XFEM to Curved Shell Surface With Arbitrary Orientations

2.1 Generation of test cases

In the previous implementation of the 2D XFEM formulation in VAST [1, 2], it was assumed that the cracked plate was flat and resided in the global X-Y plane. All the test cases generated in the previous phase of XFEM development also had these restrictions. One of the requirements for the present phase was to remove these limitations.

In order to test the XFEM capability in VAST for treating fracture mechanics problems involving curved shell geometry with arbitrary orientations, two series of test problems were prepared based on two previous test cases extensively studied in the early phases of the XFEM development. The first test case involved a plate with a 45^{o} slant edge crack and the second involved a plate with an embedded circular crack at its center. These test cases were used in the present work to generate test problems for arbitrary orientations and curved shell geometries, respectively.

In order to generate XFEM models with an arbitrary orientation, we rotated an existing XFEM model using a rotation matrix derived for arbitrarily large rotations in 3D space. For a given set of simultaneous rotations about the three axes in a global coordinate system defined as

$$\boldsymbol{\theta} = \{\theta_1 \quad \theta_2 \quad \theta_3\} \tag{1}$$

the rotation matrix can be formulated as [13]

$$\boldsymbol{R}(\boldsymbol{\theta}) = \boldsymbol{I} + \frac{\sin(\theta)}{\theta}\boldsymbol{S}(\boldsymbol{\theta}) + \frac{1-\cos(\theta)}{\theta^2}\boldsymbol{S}(\boldsymbol{\theta})\boldsymbol{S}(\boldsymbol{\theta}) \tag{2}$$

where θ is the norm of the vector of rotations and $\boldsymbol{S}(\boldsymbol{\theta})$ is a skew-symmetric matrix defined as

$$\theta = (\boldsymbol{\theta}^T\boldsymbol{\theta})^{1/2} \quad \text{and} \quad \boldsymbol{S}(\boldsymbol{\theta}) = \begin{bmatrix} 0 & -\theta_3 & \theta_2 \\ \theta_3 & 0 & -\theta_1 \\ -\theta_2 & \theta_1 & 0 \end{bmatrix} \tag{3}$$

The advantages of this rotation matrix include that (a) it is independent of the sequence of the three rotations about the global axes and (b) it preserves geometric dimensions of the original model. To obtain the rotated model, the same rotation matrix is applied to the global coordinates of nodes and crack direction vector at the crack tip. The original XFEM model involving a 45^{0} slant edge crack and some of the rotated models are shown in Figure 1 and the complete list of rotated models generated in the present work is given in Table 1. This set of rotated models included all the typical orientations to be anticipated in practical problems.

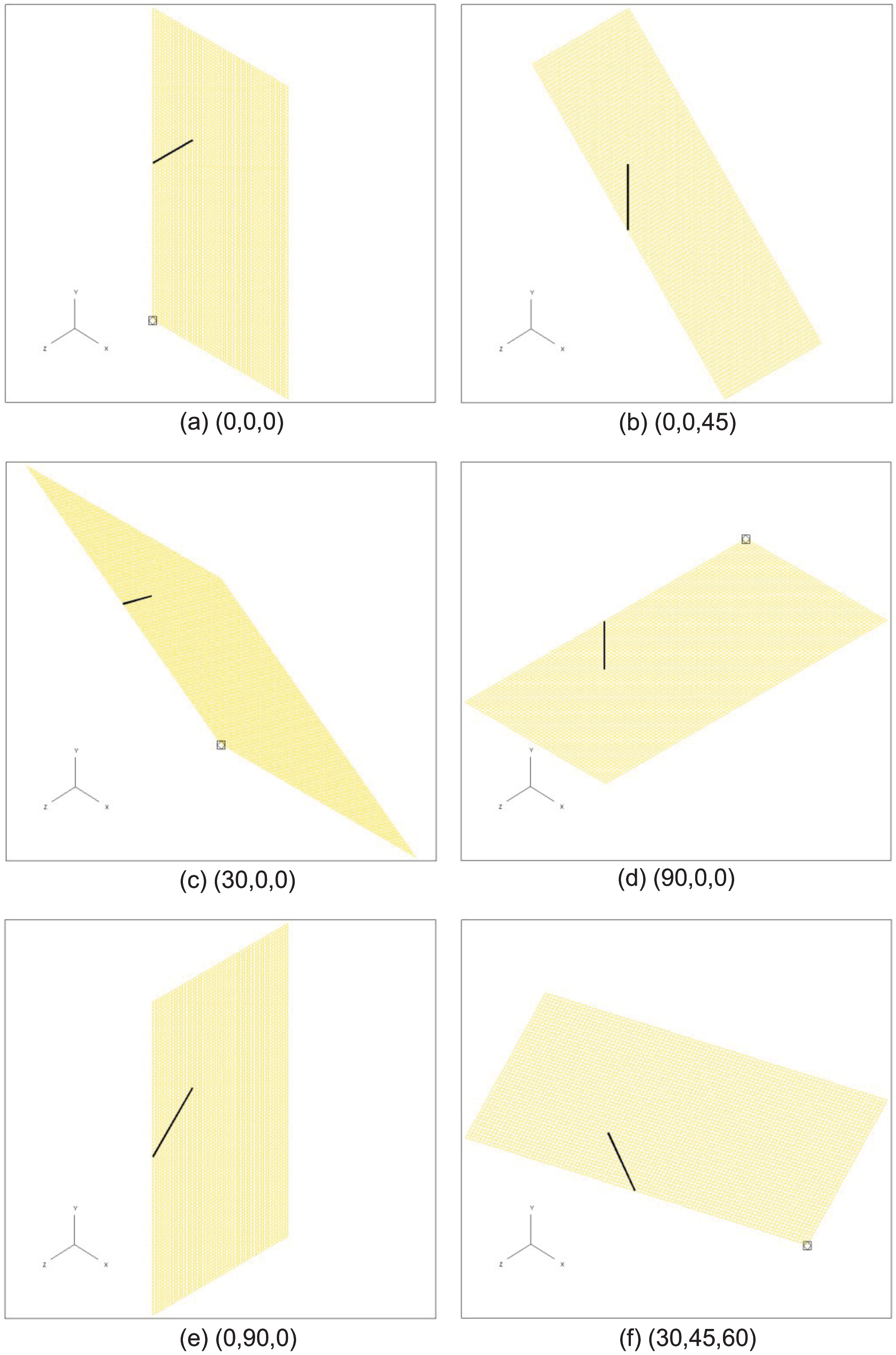


Figure 1: Rotated XFEM models for a plate with a 45^o slanted edge crack.

Table 1: Stress intensity factors obtained using the original and rotated XFEM models for the test case involving a 45^o slanted edge crack.

Rotation Angles (degree)	Direct Approach		Interaction Domain Integration	
	K_I	K_{II}	K_I	K_{II}
0,0,0	1.86252	0.891734	1.91204	0.920699
0,0,45	1.86250	0.891799	1.91228	0.920818
30,0,0	1.86240	0.891710	1.91199	0.920680
60,0,0	1.86249	0.891736	1.91200	0.920618
90,0,0	1.86252	0.891734	1.91204	0.920699
0,30,0	1.86245	0.891745	1.91205	0.920702
0,60,0	1.86242	0.891749	1.91205	0.920703
0,90,0	1.86252	0.891734	1.91204	0.920699
30,30,30	1.86307	0.891947	1.91230	0.920856
30,45,60	1.86187	0.891471	1.91168	0.920497

In order to generate test cases for curved XFEM element, we rolled the originally flat model of a center cracked plate (shown in Figure 3(a)) about either the global x- or y-direction to generate cylindrical surfaces of different curvature. Figure 2 indicates the procedure for rolling the model about the y-axis (pointing towards the reader). Assuming that the original length of the model in the x-direction is L and the central angle of the circular arc after application of curvature is ϕ, the radius of the arc can be readily obtained as

$$R = L / \phi \tag{4}$$

It should be noted that the original length of the model is preserved. For an arbitrary point in the model the central angle can be computed from the radius and the x-coordinate of the node as

$$\beta = x / R \tag{5}$$

The nodal coordinates should then be updated as

$$X = R \sin \beta \quad \text{and} \quad Z = R(1 - \cos \beta) \tag{6}$$

Models with curvature in the other direction can be generated using the same method.

Figure 3 gives a few examples of the curved XFEM models generated using the above procedure, including models with different levels of curvature about either x- or y-directions. The complete list of the curved models utilized in the present verification can be found in Table 2 below. The influences of the curvature on the predicted stress intensity factors evaluated either directly from the XFEM formulation or indirectly from interaction domain integration will be discussed later.

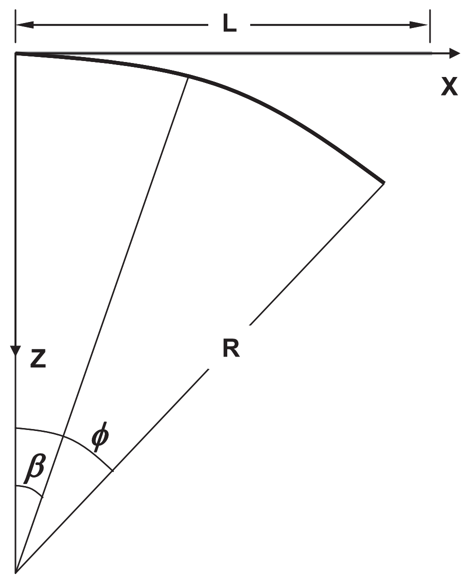


Figure 2: Generation of curved XFEM models.

Table 2: Stress intensity factors obtained using the original and curved XFEM models for the test case involving an embedded circular crack.

Central Angle (degree)	Method	Crack Tip 1		Crack Tip 2	
		K_I	K_{II}	K_I	K_{II}
0 (original, flat)	Direct	0.097285	0.957221	1.33505	-0.233623
	J-Integral	0.086048	0.900288	1.34435	-0.251316
15 (about y-axis)	Direct	0.098500	0.957746	1.33613	-0.234246
	J-Integral	0.087279	0.900820	1.34541	-0.251912
30 (about y-axis)	Direct	0.100286	0.958876	1.33814	-0.235283
	J-Integral	0.088932	0.901742	1.34747	-0.252859
45 (about y-axis)	Direct	0.102446	0.959960	1.33996	-0.236471
	J-Integral	0.091165	0.902952	1.34927	-0.253997
60 (about y-axis)	Direct	0.103455	0.961164	1.34157	-0.237567
	J-Integral	0.091943	0.903894	1.35087	-0.255033
5 (about x-axis)	Direct	0.108728	0.964846	1.34732	-0.237445
	J-Integral	0.111820	0.936686	1.29848	-0.249728
10 (about x-axis)	Direct	0.137536	0.991139	1.38560	-0.246671
	J-Integral	0.192238	1.068310	1.12604	-0.239894
20 (about x-axis)	Direct	0.220992	1.053690	1.47359	-0.269559
	J-Integral	0.479432	1.552030	0.39599	-0.185597
30 (about x-axis)	Direct	0.311243	1.089770	1.52177	-0.285089
	J-Integral	0.899191	2.268110	-0.86812	-0.078830

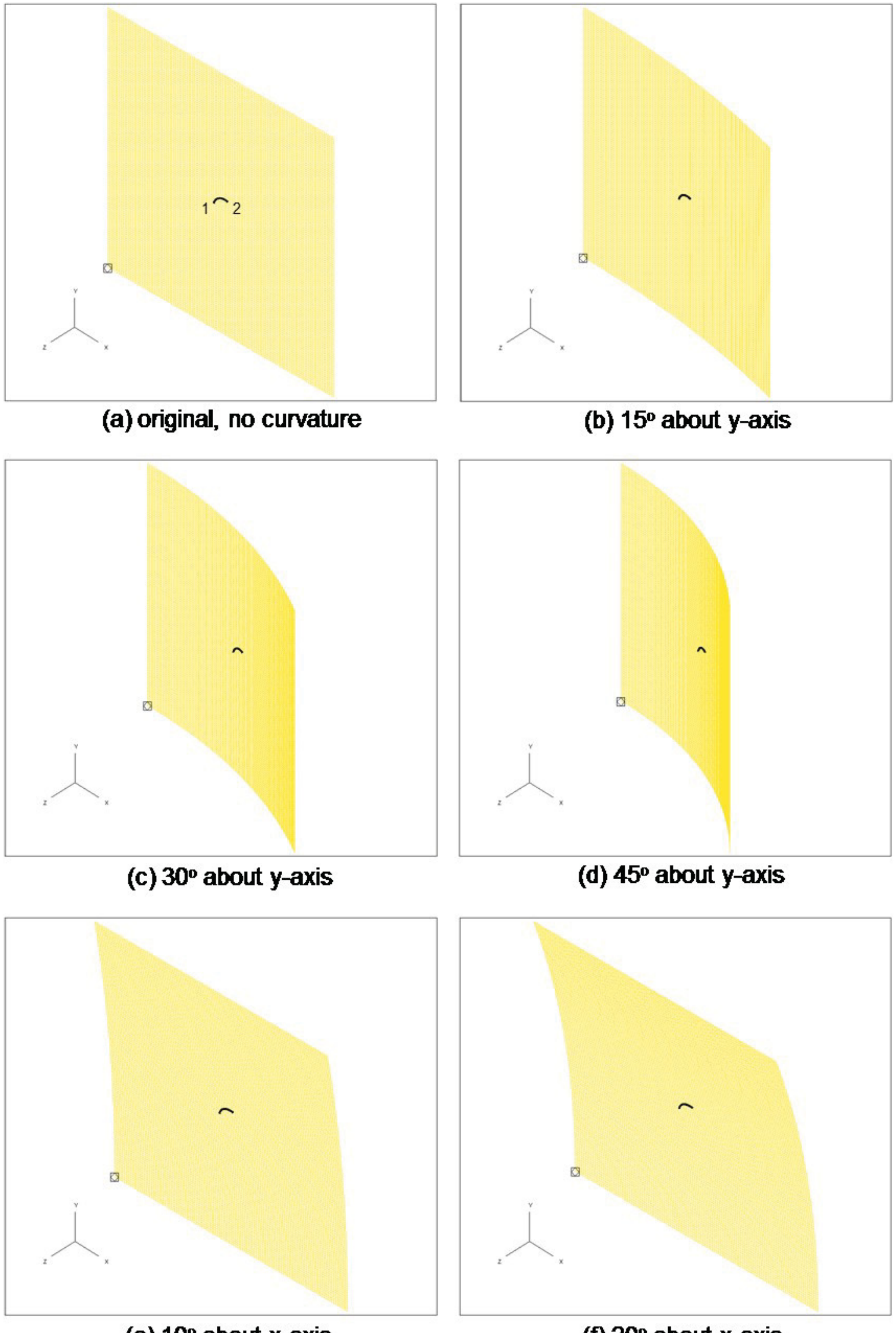


Figure 3: Curved XFEM models for a plate with a circular embedded crack.

2.2 Modifications to the XFEM fracture element

When the original VAST implementation of the 2D XFEM element, as presented in Ref [1, 2], was utilized to solve the test cases involving arbitrary orientations, it was discovered that VAST crashed in decomposition of the global stiffness matrix for models in the x-z and y-z planes due to insufficient constraints. A careful examination of the XFEM theory and the VAST source code indicated that the singularity problem was related to the insufficient constraints applied to the enrichment degrees of freedom associated with the Heaviside function $H(\mathbf{x})$ defined by

$$\boldsymbol{u}_H(\boldsymbol{x}) = H(\boldsymbol{x})\boldsymbol{a} \tag{7}$$

where $\boldsymbol{a}$ contains nodal components of in-plane displacements. Because the XFEM element was assumed to be planar and always in the global x-y plane in the previous implementation of the 2D XFEM formulation in VAST [1, 2], the z-component of the vector $\boldsymbol{a}$ was constrained at the element level to eliminate the rank deficiency in the element stiffness matrix. Obviously, this simple treatment needed to be generalized for arbitrarily oriented elements.

Further consideration indicated that the constraint condition discussed above simply required that the normal component of the enriched displacement vector $\boldsymbol{a}$ equaled zero. This condition can be expressed mathematically as

$$a_n = \begin{bmatrix} n_1 & n_2 & n_3 \end{bmatrix} \begin{bmatrix} a_1 \\ a_2 \\ a_3 \end{bmatrix} = \boldsymbol{n}^T \boldsymbol{a} = 0 \tag{8}$$

where $\boldsymbol{n}$ denotes the normal vector at the concerned node evaluated from the element geometry. In order to apply this generalized constraint to the finite element system, we noticed that it is in the form of a standard multi-point constraint equation. As a result, it can be enforced using the penalty method, just like the regular multi-point constraints treated in VAST. Defining a penalty function as

$$\Pi = \frac{\bar{k}}{2} \boldsymbol{a}^T \boldsymbol{n} \otimes \boldsymbol{n}\, \boldsymbol{a} \tag{9}$$

where $\bar{k}$ denotes the average of the diagonal terms in the element stiffness matrix before the application of the constraint conditions, the penalty stiffness matrix can be obtained as

$$\boldsymbol{K} = \bar{k}\, \boldsymbol{n} \otimes \boldsymbol{n} = \bar{k} \begin{bmatrix} n_1 n_1 & n_1 n_2 & n_1 n_3 \\ n_2 n_1 & n_2 n_2 & n_2 n_3 \\ n_3 n_1 & n_3 n_2 & n_3 n_3 \end{bmatrix} \tag{10}$$

2.3 Verification of the modified XFEM element

The modified 2D XFEM element was first verified using the arbitrarily oriented test cases of a plate containing a 45^0 slant edge crack and the stress intensity factors obtained from all test cases using both the direct approach and domain integration are compared with the results from the original model in Table 1. The results from all models are almost identical. The small differences between the solutions were believed to be caused by the limited number of significant digits used for the nodal coordinates in the input data files for VAST, which caused small inconsistences between the nodal coordinates in the original and oriented models. These results confirmed that the modified 2D XFEM element behaved properly for elements with arbitrary orientations.

The test cases for curved XFEM models can be divided into two groups based on the directions of the applied curvature. Models in the first group were created by rolling the original model about the y-axis. Since in the original model, the uniform uni-axial tensile stress field was applied along the global y-axis, the test cases in this group still underwent purely in-plane deformations. Under this circumstance, both the direct and the interaction domain integration methods produced very consistent solutions as summarized in lines 3 to 10 in Table 2, where slightly increased stress intensity factors were obtained for increased curvature. These results indicated that the modified 2D XFEM element works correctly for curved element geometry as long as it is under membrane deformation.

For the second group of test cases which were generated by rolling the original model about the x-axis, the application of uniform edges forces in the y-direction resulted in combined membrane and bending deformations. In this case, the stress intensity factors obtained from the direct approach (lines 11, 13, 15, 17 in Table 2) were stable, but slightly more sensitive to the curvature of the model relative to the first group of the test cases discussed in previous paragraph. However, due to the lack of analytical solutions, no direct evaluation of the accuracy of numerical solutions is possible.

The results from the domain integration (lines 12, 14, 16, 18 in Table 2) were found to be extremely sensitive to the curvature in the model. In particular, with the increase of curvature, the stress intensity factors for crack tip 1 increased rapidly, whereas the stress intensity factors for crack tip 2 decreased. These results are clearly not reasonable. This is probably because the interaction domain integration method implemented in VAST [2] is intended for in-plane deformation, which is not applicable to the mixed membrane-bending deformations in this set of test cases.

3 Extension of XFEM to Allow Crack to Pass Element Corners

In the previous implementations of the XFEM element in VAST as described in Ref [1, 2], the crack was always assumed to intersect with element edges, but not pass directly through any of the corners of the element. Although this requirement could always be satisfied by adjusting the underlying finite element mesh, it introduced some difficulties to the full automation of the calculation for crack-mesh interaction. In order to eliminate this limitation, the XFEM capability in VAST was extended to permit the crack to pass through one or more corners of an element. For each corner, six possibilities may occur as indicated in Figure 4 below. In order to accommodate all these situations, the algorithms for treating crack-element interactions were generalized, where special considerations were taken in defining nodal values of discontinuous enrichment functions at the nodes located on the crack line.

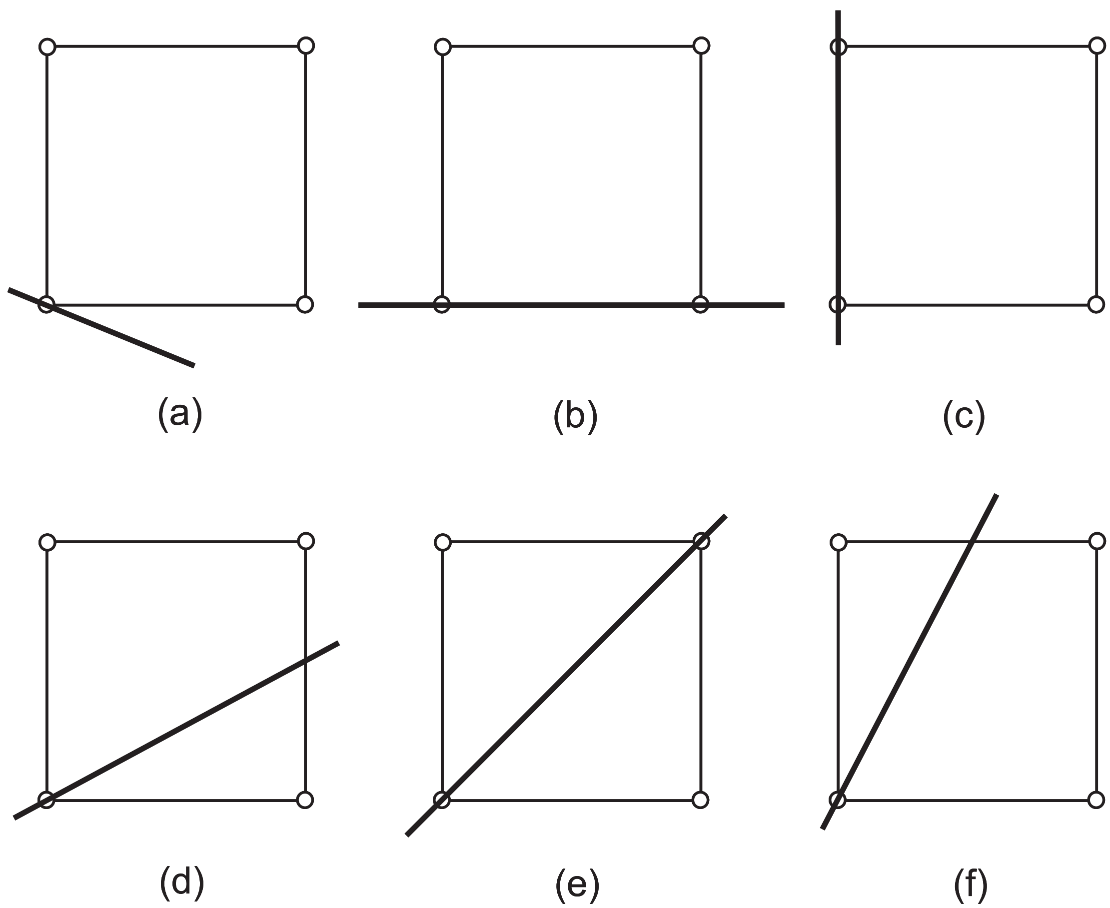


Figure 4: Cases for crack passing one or more corners of a XFEM element.

To verify the modified VAST program which permits a crack to pass through corners of a XFEM element, we reconsidered the test example involving a center cracked square plate subjected to a uniform stress field as shown in Figure 5. The in-plane dimension of the square plate was W=10.0 and the half crack length, a, was taken as 1.0. In this example, we attempted to obtain the K_I and K_{II} stress intensity factors as a function of the angle β by using two fixed uniform underlying meshes including a 40×40 coarse mesh and a 200×200 fine mesh. The XFEM models based on both the coarse and fine meshes for β=0^O, 30^O and 45^O are given in Figures 6-8, where the complete models and the local details around the crack are both displayed. It should be noticed that all six configurations of crack-element interaction shown in Figure 4 are covered in these models. In particular, cases (b) and (c) were considered in models for β=0^O, cases (d) and (f) were included in models for β=30^O, and cases (a) and (e) appeared in models for β=45^O.

For the given loading condition and crack geometry, analytical solutions of stress intensity factors for an infinite plate are available as [3]

$$\begin{aligned} K_I &= \sigma\sqrt{\pi a}\cos^2(\beta) \\ K_{II} &= \sigma\sqrt{\pi a}\cos(\beta)\sin(\beta) \end{aligned} \tag{11}$$

The XFEM results obtained using different underlying meshes are compared with the analytical solutions in Table 3. For all three β values considered in this study, the mode I and mode II stress intensity factors predicted by VAST using the direct approach and domain interaction integration are in good agreement with the analytical solutions. Comparing to the results from the direct approach, the stress intensity factors generated by domain integration are significantly less sensitive to the finite element model.

Table 3: Test results for XFEM models with crack passing through element nodes.

Solution Method	**K_I**		**K_{II}**	
	Direct	**Domain Integral**	**Direct**	**Domain Integral**
(a) Orientation angle β=0^O				
XFEM coarse mesh	6.3985	7.4993	0.0000	0.0000
XFEM fine mesh	7.2984	7.4848	0.0000	0.0000
Analytical	7.0898	7.0898	0.0000	0.0000
(b) Orientation angle β=30^O				
XFEM coarse mesh	5.0572	5.6657	2.8849	3.2607
XFEM fine mesh	5.5865	5.6061	3.1663	3.1961
Analytical	5.3174	5.3174	3.0700	3.0700
(c) Orientation angle β=45^O				
XFEM coarse mesh	3.2158	3.3469	2.4980	2.9956
XFEM fine mesh	3.5294	3.3556	3.0533	3.1008
Analytical	3.5449	3.5449	3.5449	3.5449

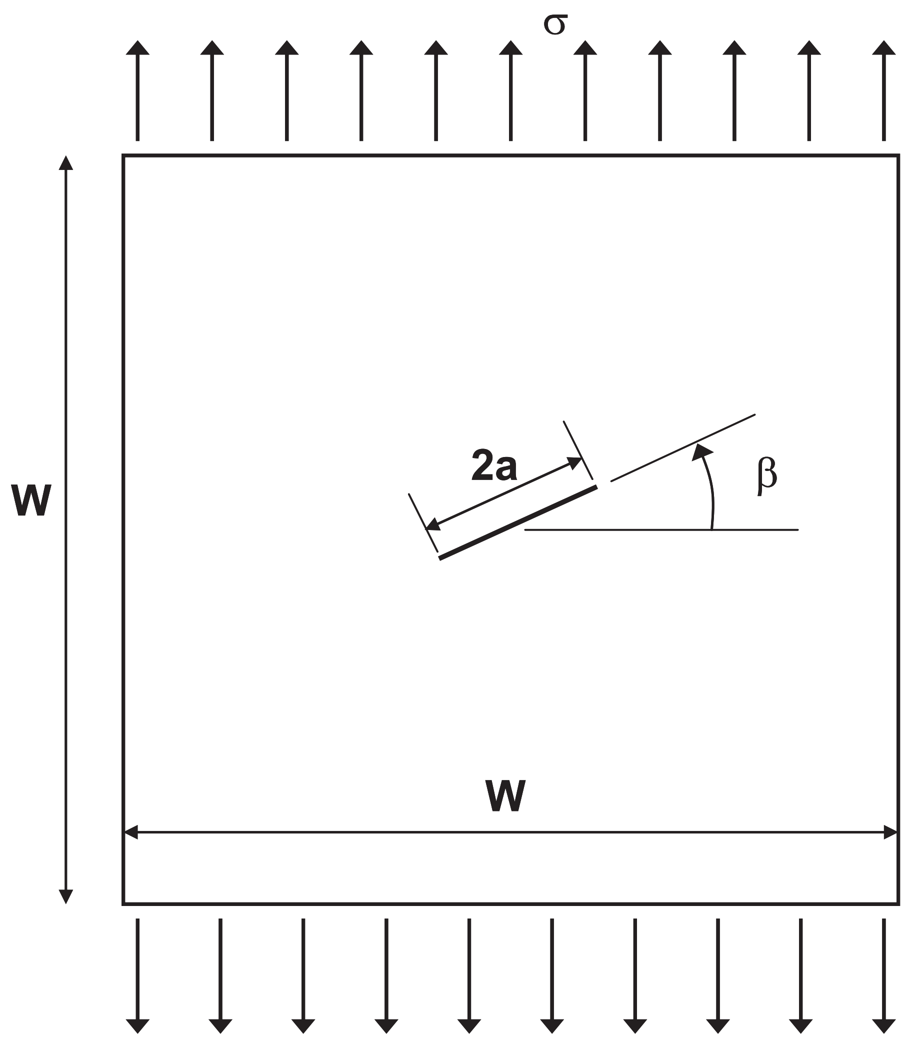
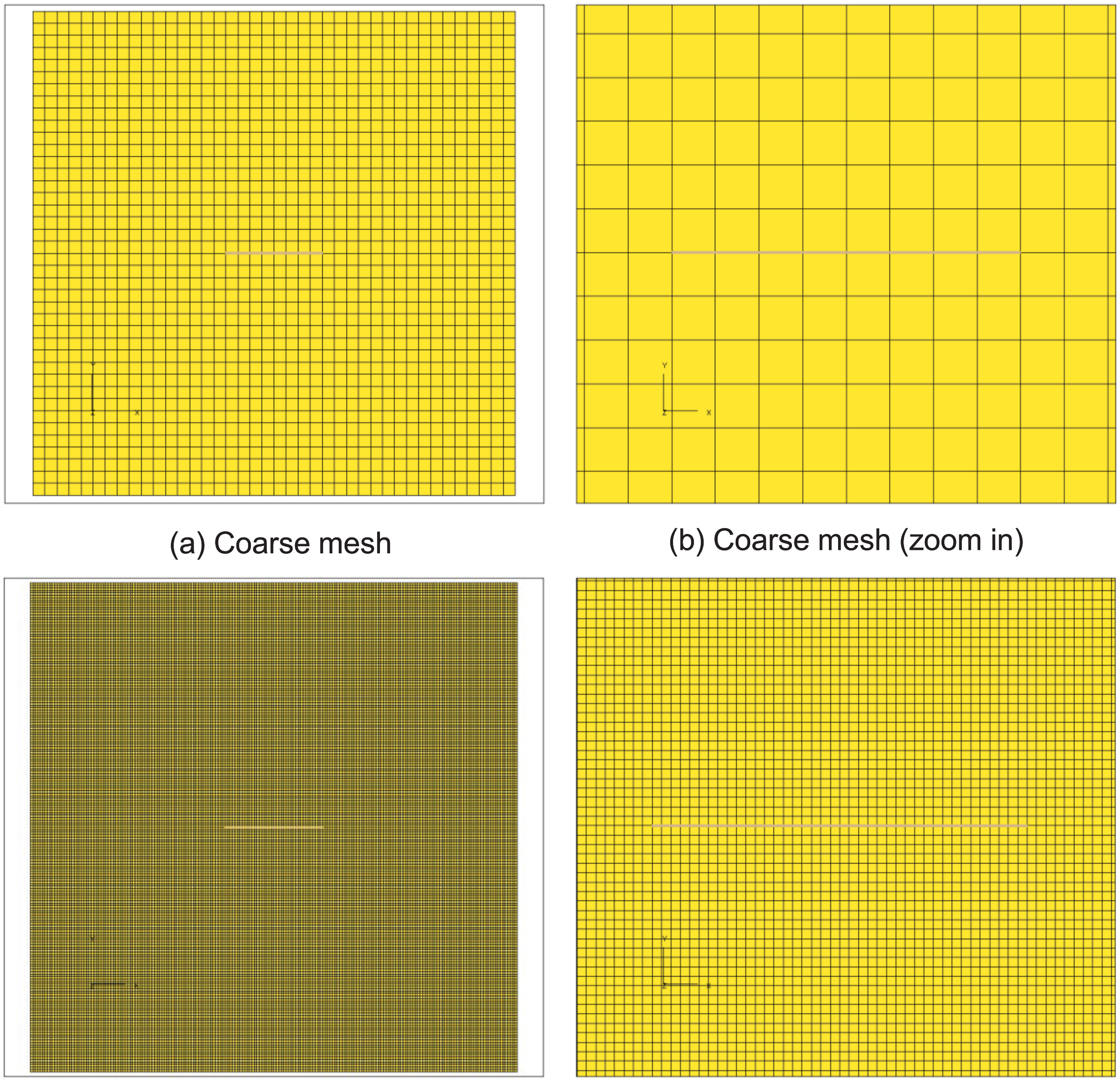


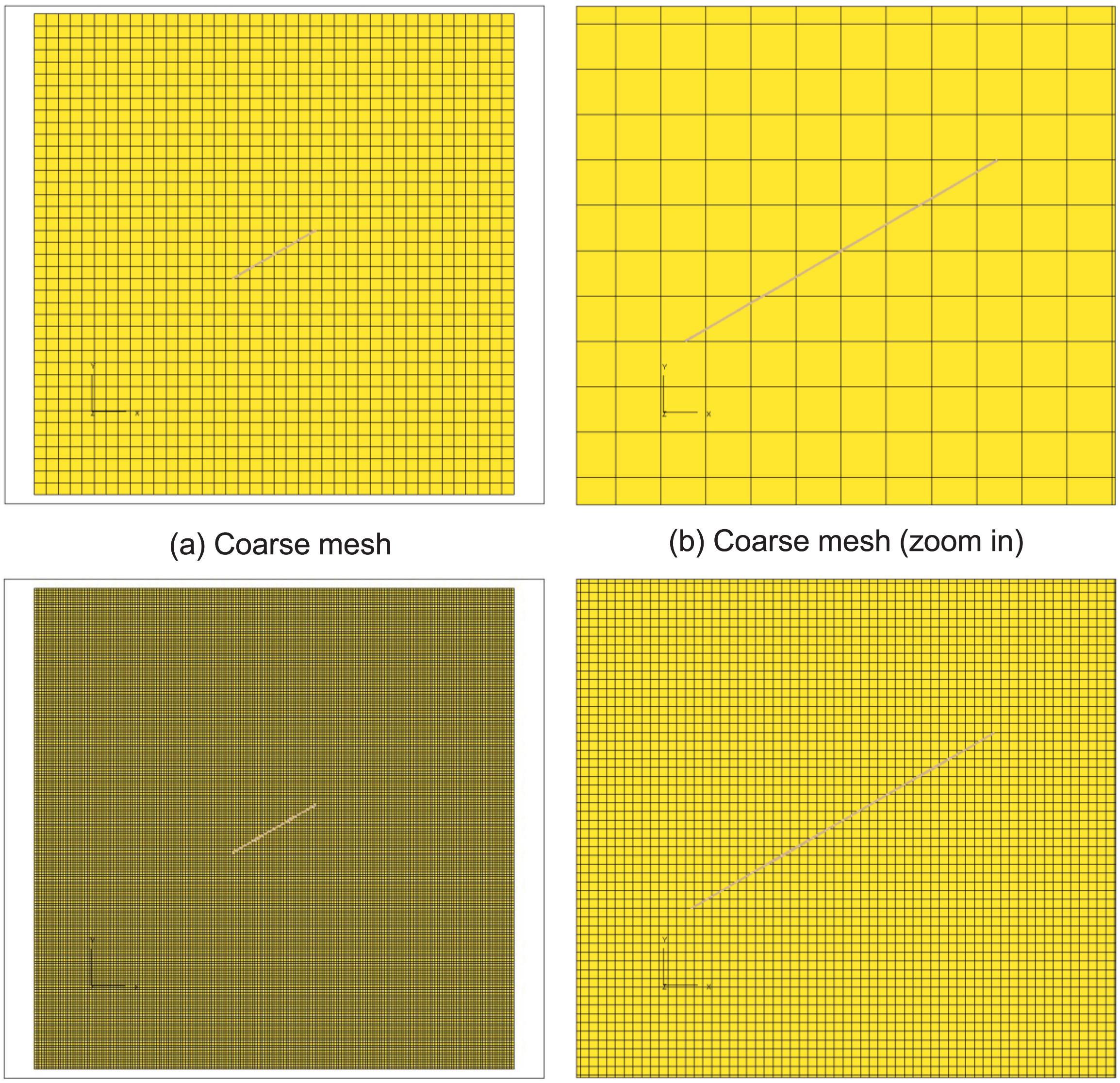
Figure 5: Plate with angled center crack.



(a) Coarse mesh (b) Coarse mesh (zoom in)

(c) Fine mesh (d) Fine mesh (zoom in)

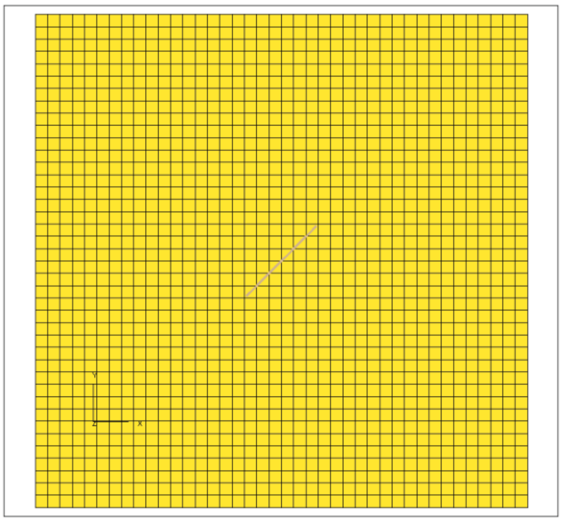
Figure 6: XFEM models of different levels of refinement for a center cracked plate with a crack length a=1 and an orientation angle $\beta=0^o$.



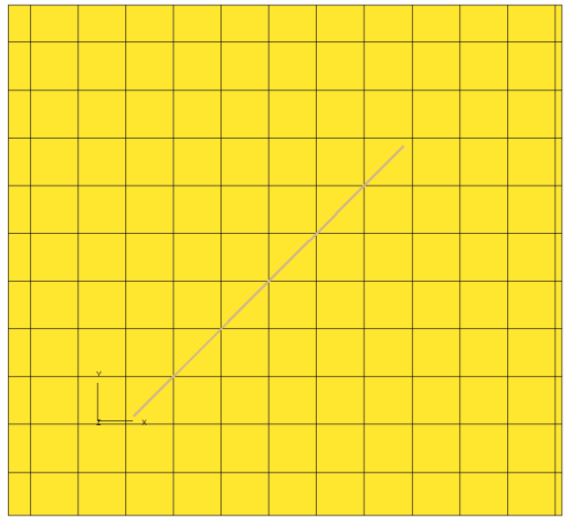
(a) Coarse mesh (b) Coarse mesh (zoom in)

(c) Fine mesh (d) Fine mesh (zoom in)

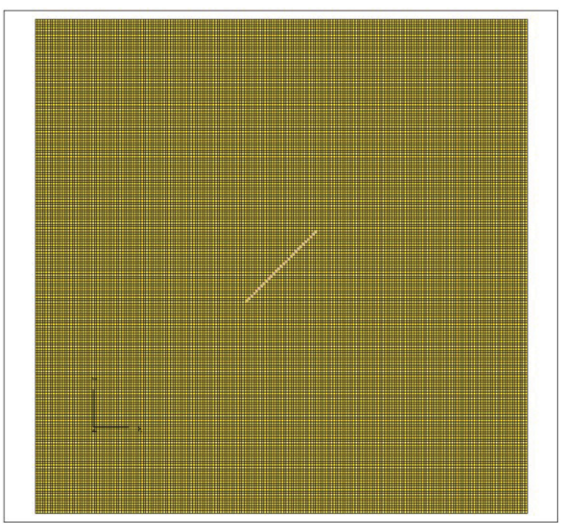
Figure 7: XFEM models of different levels of refinement for a center cracked plate with a crack length a=1 and an orientation angle β=30^{o}.



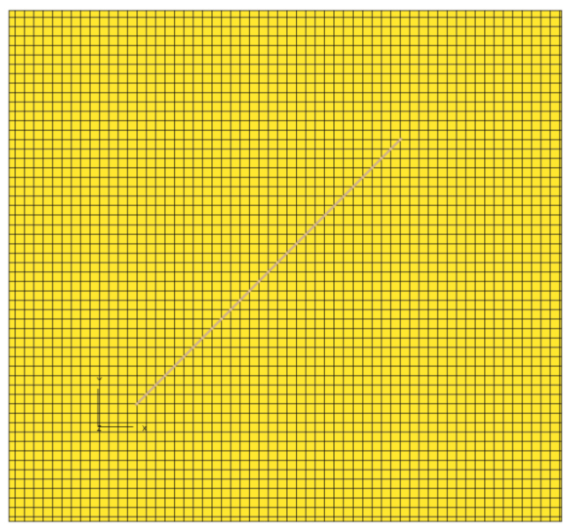
(a) Coarse mesh



(b) Coarse mesh (zoom in)



(c) Fine mesh



(d) Fine mesh (zoom in)

Figure 8: XFEM models of different levels of refinement for a center cracked plate with a crack length a=1 and an orientation angle β=45^{o}.

4 Implementation and Verification of XFEM Capability in TRIDENT

The recent work performed on TRIDENT to accommodate the newly added XFEM fracture analysis capability in VAST is described in this chapter.

At the present time, this XFEM element formulation was only implemented into a four node fracture element in VAST. This four node fracture element with XFEM enrichment has been named element type IEC=68. In a previous version of TRIDENT, the ability to perform fracture analysis was incorporated based on the four node conventional fracture element IEC=18. This conventional fracture element was formulated using the enriched solid element formulation and is referred as the element with regular enrichment in this report.

As far as TRIDENT is concerned, there is only one four node fracture element, which can have either the regular enrichment or XFEM enrichment formulations. When the element is exported to VAST, it is transparently defined as either element type IEC=18 or IEC=68 depending on the type of the enrichment field defined for this element.

The pre-processing in TRIDENT associated with fracture analysis using the fracture elements is described in Section 4.1. The post-processing in TRIDENT associated with fracture analysis using the four node fracture element is described in Section 4.2. Some verification results are discussed in Section 4.3.

4.1 Pre-processing in TRIDENT

Here, pre-processing is defined as the generation of the four node fracture element data, including connectivity, material and geometric properties, and special enrichment codes unique to each type of fracture element – as described in the VAST User's Manual [4].

The very first task involved in the generation of finite element models for fracture mechanics analysis is to create fracture elements. One of the most convenient ways to do this is to convert the regular four-node quad shell elements around the crack to four node fracture elements. This can be accomplished by following either of the two paths given below:

Generate/Modify → Elements → Options → Convert Type → To Fracture or

Generate/Modify → Crack Lines and Tips → Convert Elements → To Fracture.

In this operation, the user needs to define the shell elements to be converted and take either of the two paths described above. The current active elements will be converted. A typical example of such conversion is shown in Figure 9.

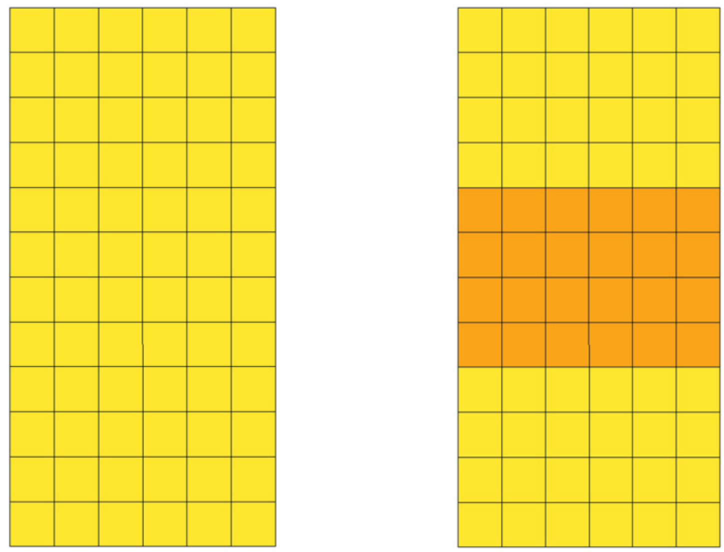


Figure 9: Creation of fracture elements by converting regular shell elements.

Essential to fracture analysis in TRIDENT and VAST is the use of crack lines and crack tips. A crack line in TRIDENT/VAST is an entity comprising a straight line segment formed by two nodes. A crack tip in TRIDENT/VAST is an entity comprising a single node – this node being an existing crack line node at one end of the crack. For fracture elements with regular enrichment (IEC=18), crack lines and tips must be defined using existing element connectivity nodes in the structural model. For XFEM enrichment, crack lines and tips must be defined using unique nodes – that is, nodes not part of the existing element connectivity. Examples of the two ways of representing crack lines and tips are provided in Figure 10.

Crack lines are created, deleted and initialized under the ***Crack Lines*** menu accessible through:

Generate/Modify → Crack Lines and Tips → Crack Lines

For regular fracture elements (IEC=18), the crack lines are defined by clicking on the structural nodes along the crack. For the XFEM fracture elements, the crack does not normally coincide with element edges. In this case, the crack lines can be specified using either pre-defined pseudo-nodes or pre-defined line or circular arc primitives. To define the pseudo-nodes, use

Generate/Modify → Nodes → Create

To define line or circular arc primitive entities, use

Generate/Modify → Primitives → Create

For detailed instructions on how to create primitives, please refer to TRIDENT User's Manual [5]. Once the primitive is selected, a group of pseudo-nodes are automatically generated and ready to be used for crack lines definitions. Because at the present time, only a single line

primitive is permitted in crack line definitions and in TRIDENT linear primitives are smooth, the option for using pseudo-nodes must be adopted for defining kinked cracks.

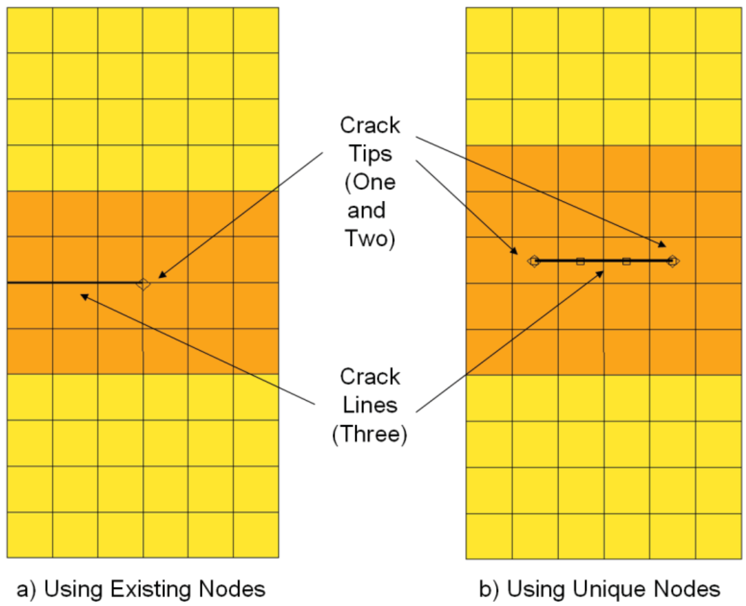


Figure 10: Examples for definition of crack lines and tips for regular and XFEM fracture elements.

Crack tips are created, deleted and initialized under the ***Crack Tips*** menu which can be accessed through

Generate/Modify → Crack Lines and Tips → Crack Tips

To define a crack tip, simply click on the nodes at the end of the crack line. For embedded cracks, two crack tips need to be defined.

For fracture analysis with the regular fracture element enrichment (IEC=18), the crack must be explicitly modeled along element edges. This means that the model must be split along the crack, up to the crack tip. This otherwise tedious operation can be done automatically with TRIDENT under the ***Split Model*** menu through path

Generate/Modify → Crack Lines and Tips → Split Model

To split model, the user needs to select elements on one side of the crack. The result of such an operation is shown in Figure 11. This operation can be reversed by using the **Seam Model** menu on the same page. It should be noted that this operation is not required for the XFEM elements.

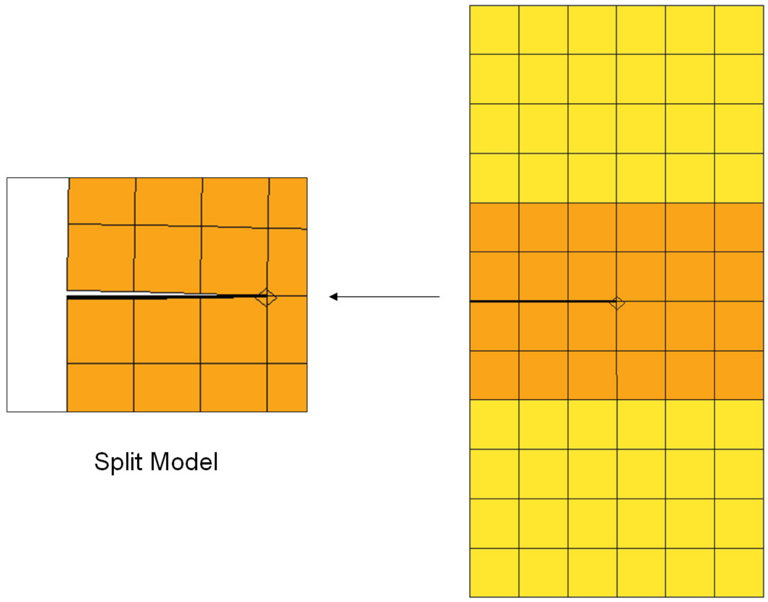


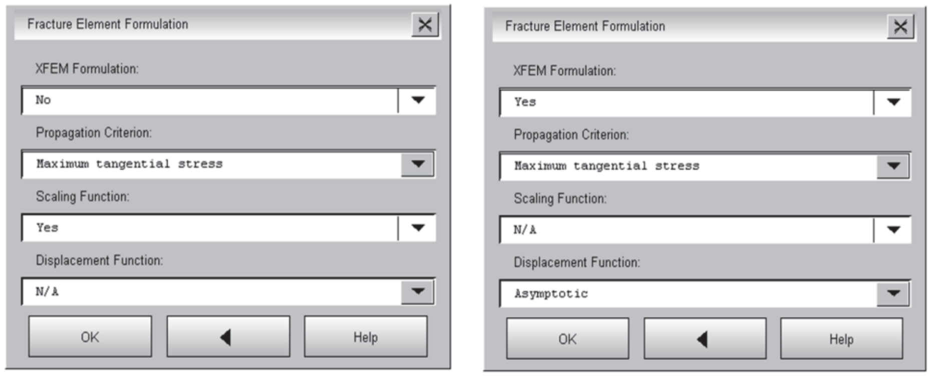
Figure 11: Using the split model operation in TRIDENT.

Using either of the two models shown in Figures 10, the user then proceeds to assign the fracture element enrichment formulation data. This can be done under the 2-D 4 Node Elements menu accessible through

Generate/Modify → Crack Lines and Tips → Fracture Element Formulation

Figure 12 (a) and (b) indicate assignment of enrichment functions for the regular and XFEM fracture elements, respectively. To assign the enrichment field for the regular fracture elements for kinked cracks, the user needs to click on the crack tip and then define the area to which the enrichment functions are to be applied.

For XFEM, the assignment of the two types of enrichment functions must be done in two separate steps. In Step 1, the asymptotic crack tip enrichment fields are assigned in the same way as in the regular fracture element. This is to click on the crack tip first and then define the area to which the crack tip enrichment functions are to be applied. In Step 2, the discontinuous enrichment function is to be assigned by defining the elements on both sides of the crack line to which the crack tip enrichment functions are not applied.



a) Regular Element Enrichment b) XFEM Element Enrichment

Figure 12: TRIDENT menu for defining fracture element enrichment formulation data.

Once the enrichment data have been assigned, data verification can be carried out using the ***Inquire / Elements*** menu in TRIDENT. Figure 13 shows the result of a verification of a regular enriched element – note in particular, the values for the edge compatibility flags IST 1/2/3/4. These flags are used to formulate scaling functions to the enrichment field to ensure displacement compatibility between the regular fracture elements and 4-noded quad shell elements [6]. Figure 14 shows the result of a verification of an XFEM enriched element – note in particular, the values for the XFEM node enrichment flags IENR 1/2/3/4, where IENR=1 indicates enrichment by the discontinuous displacement function and IENR>1 indicates enrichment by the asymptotic displacement functions associated with different crack tips. Note IENR=2 corresponds to crack tip # 1, IENR=3 corresponds to crack tip # 2, and so on.

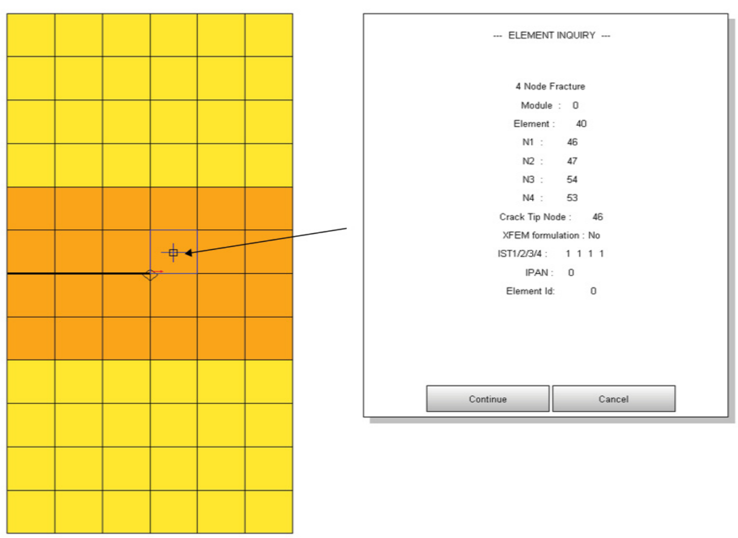


Figure 13: Verification of regular fracture element enrichment data using the inquire feature in TRIDENT.

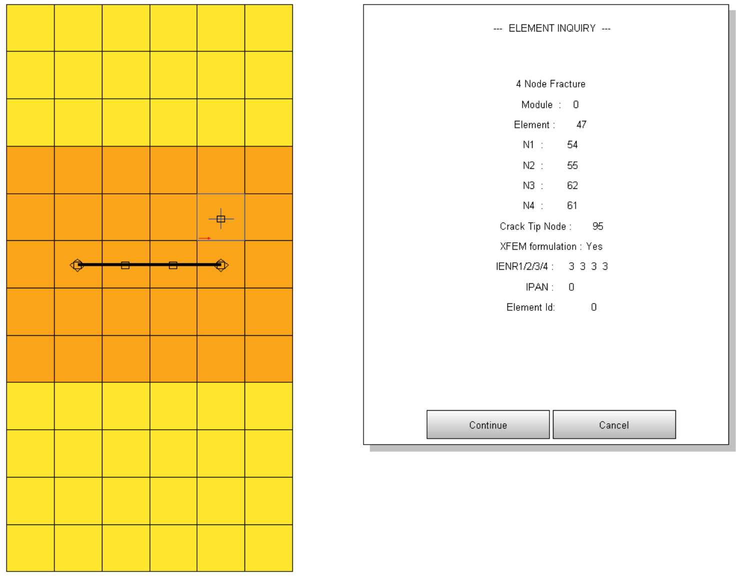


Figure 14: Verification of XFEM fracture element enrichment data using the inquire feature in TRIDENT.

Verification of XFEM node enrichment flags can be carried out using the special feature named **XFEM Enrichment Flags** in the ***Verify / Crack Tips*** menu. An example of this is shown in Figure 15.

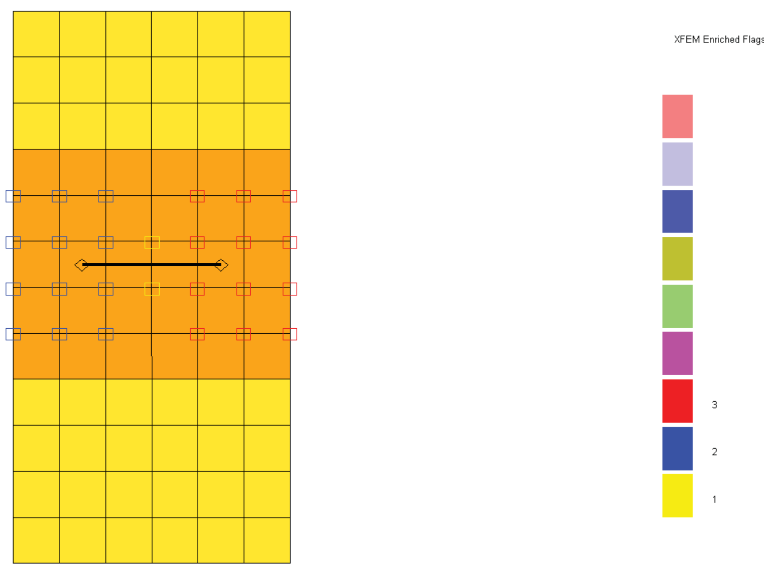


Figure 15: Verification of XFEM fracture element enrichment data using the verify feature in TRIDENT.

4.2 Post-processing in TRIDENT

Here, post-processing is defined as the presentation of nodal displacements, element stresses, and, in particular, the stress intensity factors obtained by the VAST solver. Displacements and element stresses are presented in the usual way in TRIDENT. For fracture elements, there are two stress intensity factors and the angle for crack propagation included in the stress component list, as shown in Figure 16.

Sample displacement and stress intensity plots for a fracture analysis using the regular fracture elements are provided in Figure 17. In these fracture elements, the enrichment field was formed in terms of the stress intensity factors of the crack tip(s), so the stress intensity factors were obtained directly along with the nodal displacements and could be conveniently displayed as in Figure 17(b). The displacement and stress contour plots from a typical fracture analysis using the XFEM fracture elements are provided in Figure 18. The stress plot was generated based on the average stresses in each element. As demonstrated by the example problems discussed earlier in this report, in the XFEM fracture element, accurate computation of the stress intensity factors required post-processing techniques, such as the domain interaction J-integral. As a result, the stress intensity factors cannot be conveniently reported along with the element stresses. In the present implementation, the calculated stress intensity factors are reported in the LPT file. Unfortunately, at the present time, graphical display of these stress intensity factors is not yet permitted.

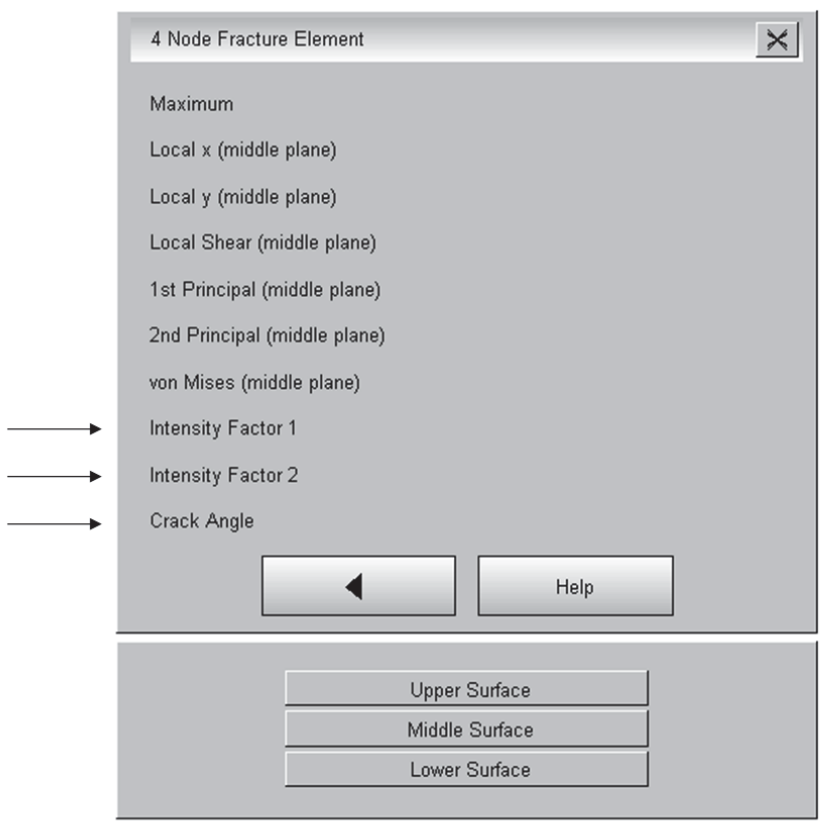
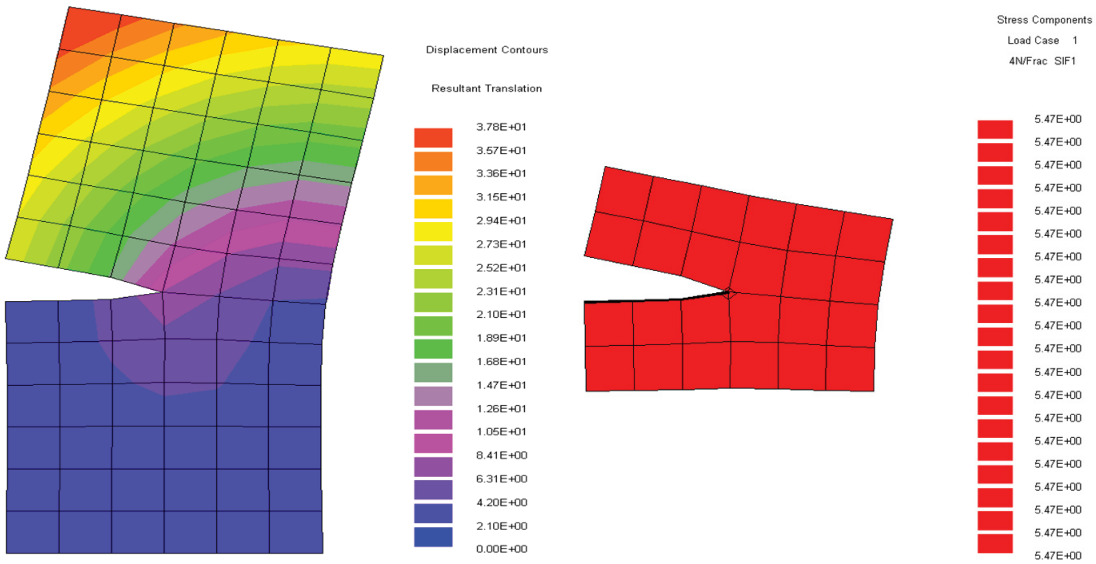
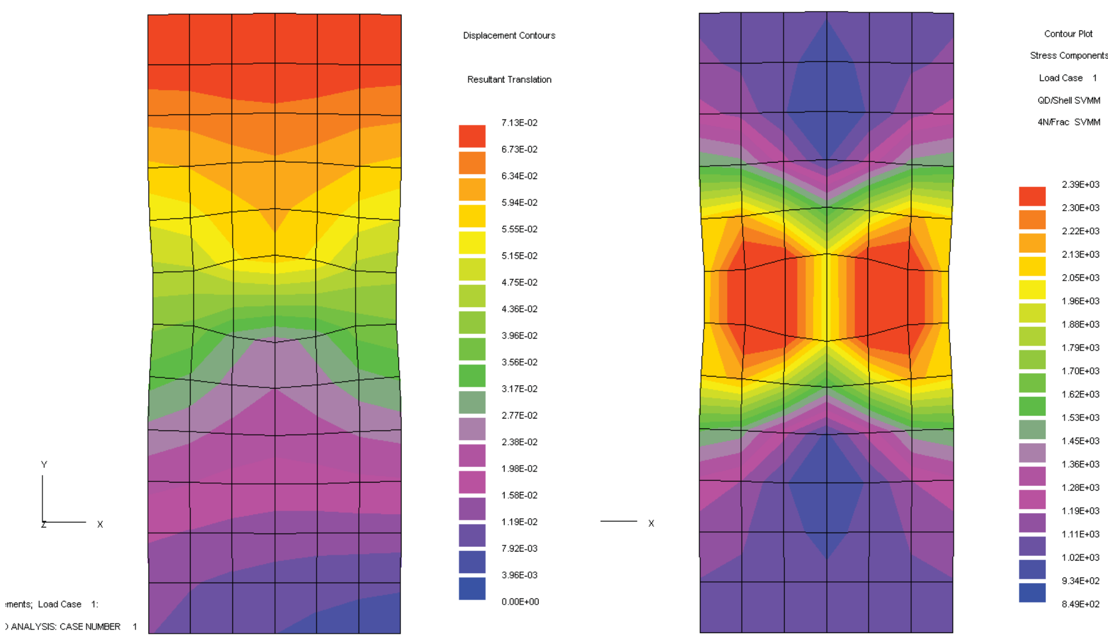


Figure 16: TRIDENT stress component list for the four node fracture element.



(a) Displacement contour (b) Stress intensity factor

Figure 17: TRIDENT generated plots from a fracture analysis using regular enriched fracture elements IEC=18.



(a) Displacement contour (b) Von Mises stress contour

Figure 18: TRIDENT generated plots from a fracture analysis using XFEM enriched fracture elements IEC=68.

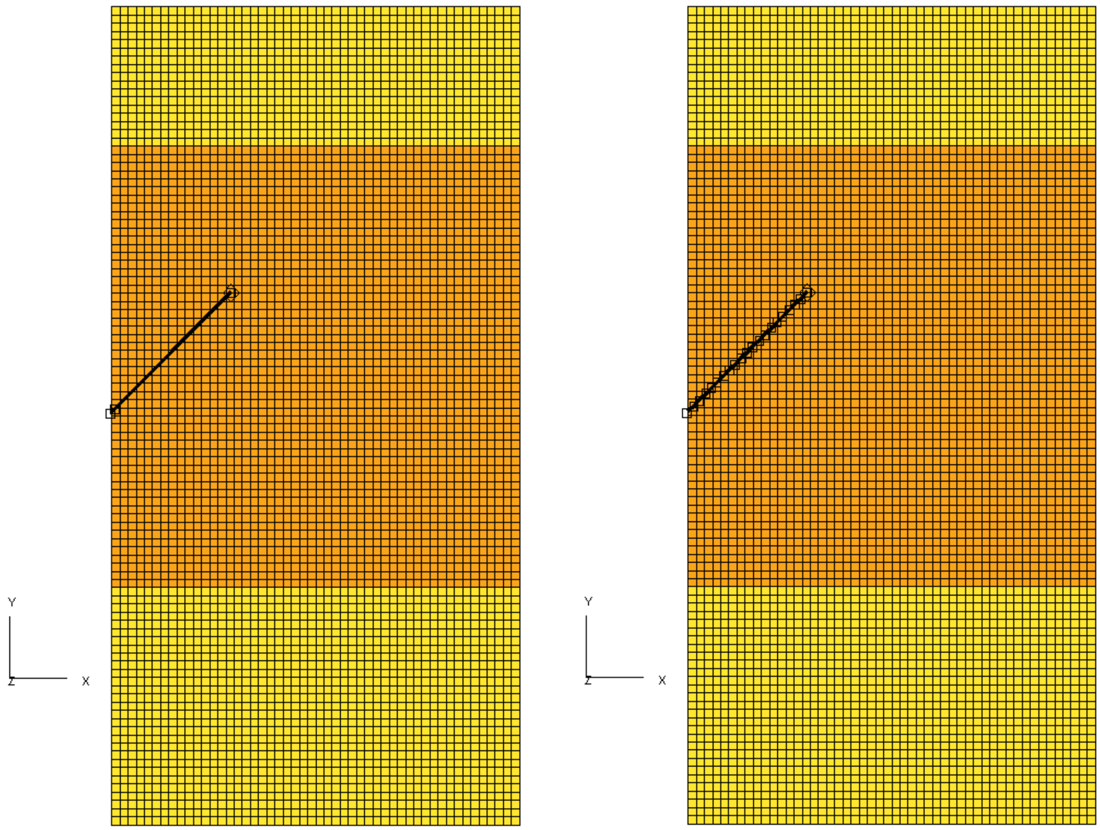
4.3 Verification of XFEM capability in TRIDENT

In addition to the relatively simple test cases described above, extensive numerical verifications of the XFEM modeling capability in TRIDENT were carried out using more complicated test problems involving mixed mode deformations, including a rectangular plate with a 45^{O} slant edge crack and square plates with straight and circular center cracks as indicated in Figures 19-21. All of these test cases have analytical solutions [3,7,8] and were previously solved using both the regular and XFEM enriched fracture elements in VAST [1,2,6,9].

In order to verify the XFEM feature in TRIDENT thoroughly, three new analyses were performed for each of these test cases. In the first analysis, the previous XFEM models, which were created during the earlier phases of the XFEM development using special pre-processors [2], were imported into TRIDENT and then, exported to VAST input data files. These input data files were then used in VAST analyses. The current results were compared with the previous ones in Table 4, where exact agreement has been found. This level of agreement confirmed that TRIDENT was able to import and export XFEM models correctly.

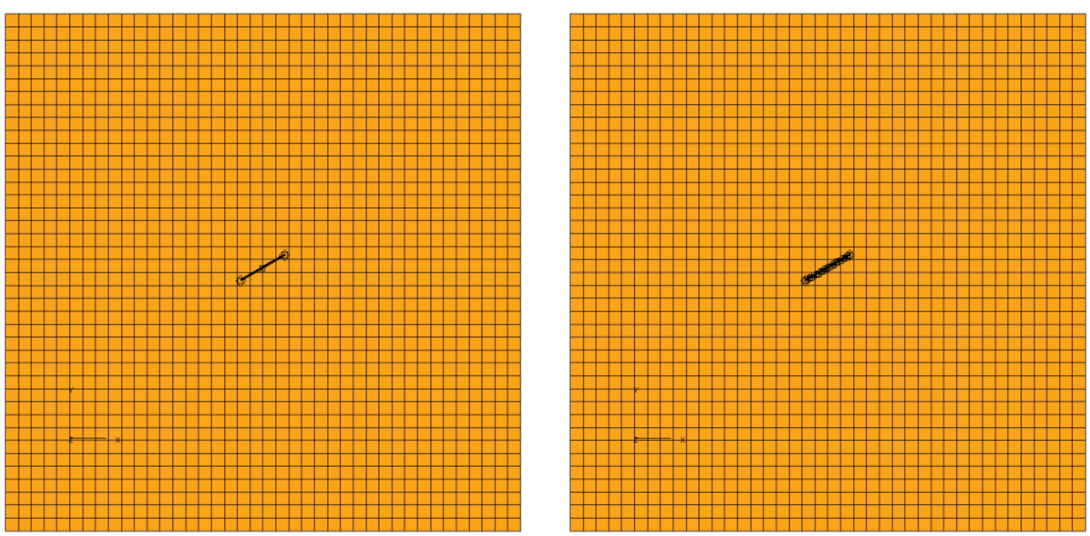
In the second and third analyses, the original XFEM models were first imported into TRIDENT, but the previously defined crack lines and crack tips were removed through initialization. The crack geometry was then defined using the TRIDENT crack modeling options described in the previous sections in this chapter. In Analysis 2, crack lines were defined through user-supplied pseudo-nodes, whereas in Analysis 3, the crack geometry was specified using line and circular arc primitives. These newly generated XFEM models are displayed in Figures 19, 20, 21 and the stress intensity factors predicted by these models are presented in Table 4. These results are in good agreement with the previous solutions. The discrepancies between the solutions for the circular crack problem obtained using different modeling options are related to the differences in the representation of the crack geometry and the areas to which the enrichment displacement functions were assigned. In TRIDENT, the density of grid points to be assigned to represent a circular arc primitive is fixed to 5^{o} per point. As a result, for the present test problem where the circular crack corresponded to a center angle of 90^{o}, a total of eighteen equally spaced grid points were generated along the crack. With this grid point density, the orientation of the crack tip local coordinate systems in the analytical solution could not be accurately represented, resulting in errors in the numerical solutions.

In order to confirm the TRIDENT XFEM capability for dealing with piece-wise linear crack geometry, we reconsidered a kinked crack problem that was solved in the previous phase of this project [2]. To generate the XFEM models using TRIDENT, we utilized the same underlying finite element mesh used for the circular crack problem and introduced the kinked crack through both pseudo-nodes and linear primitives. These XFEM models are shown in Figure 22. It should be noted that during the definition of crack lines, twenty grid points are generated for each of the line primitives. As a result, the model generated using linear primitives contained significantly more points along the crack lines than the model created through user-specified pseudo-nodes. The number of points used in crack line representation has an influence on the subdivision of the fracture elements and hence numerical integration in these elements. This eventually causes some small differences in the final results as indicated in Table 5. These results confirmed that the present XFEM capability in TRIDENT does permit kinked cracks.



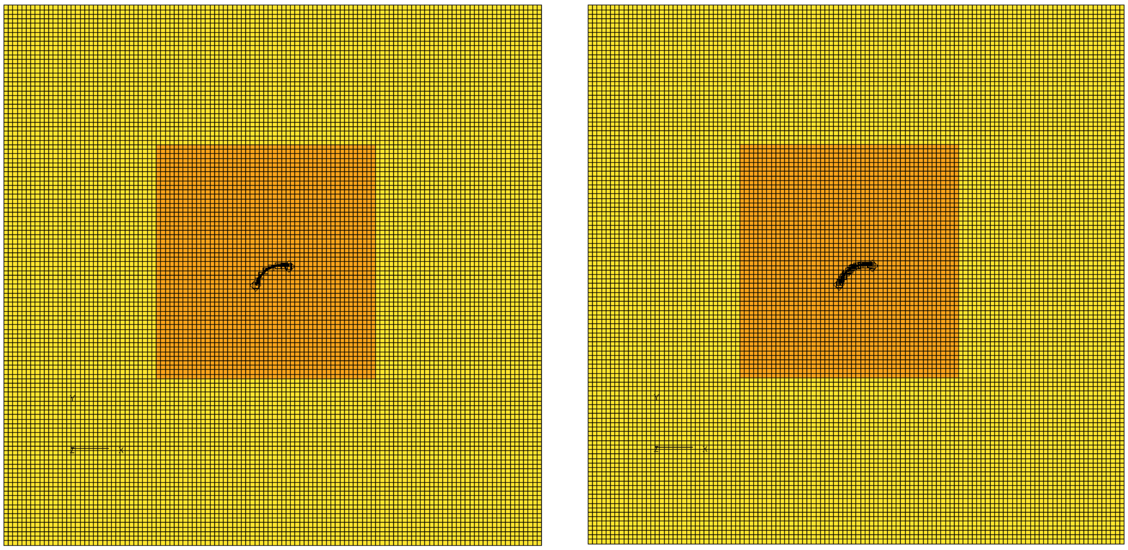
(a) Crack lines by nodes (b) Crack lines by line primitive

Figure 19: TRIDENT generated XFEM models for a plate with a 45^{O} slant edge crack.



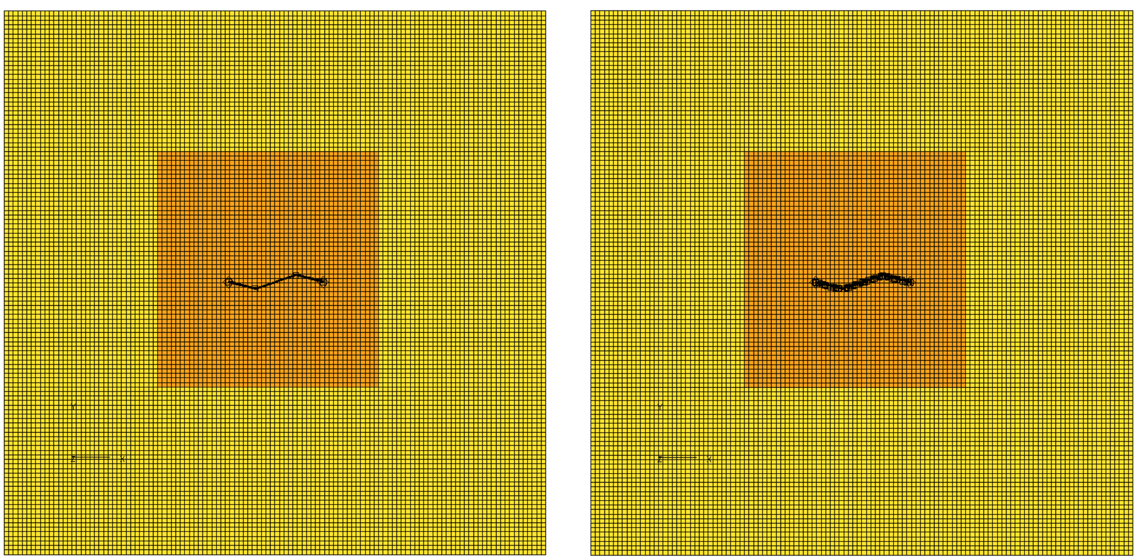
(a) Crack lines by nodes (b) Crack lines by line primitive

Figure 20: TRIDENT generated XFEM models for a plate with a 30^{O} center crack.



(a) Crack lines by nodes (b) Crack lines by circular arc primitive

Figure 21: TRIDENT generated XFEM models for a plate with a circular center crack.



(a) Crack lines by nodes (b) Crack lines by circular arc primitive

Figure 22: TRIDENT generated XFEM models for a plate with a kinked center crack.

Table 4: Comparison of stress intensity factors obtained using different XFEM models for selected test cases.

XFEM Model	Crack Tip 1		Crack Tip 2	
	K_I	K_{II}	K_I	K_{II}
(1) Plate with a 45^O slant edge crack (see Figure 19)				
Original	1.914289	0.922162		
Analysis 1	1.914289	0.922162		
Analysis 2	1.913921	0.922430		
Analysis 3	1.915764	0.923607		
(2) Plate with a 30^O center crack (see Figure 20)				
Original	3.817490	2.262520	3.950600	2.157790
Analysis 1	3.817490	2.262520	3.950600	2.157790
Analysis 2	3.913864	2.218538	3.937438	2.197379
Analysis 3	3.907727	2.220811	3.931220	2.196822
(3) Plate with a circular center crack (see Figure 21)				
Original	0.0882320	0.8972871	1.354480	-0.2645029
Analysis 1	0.0882320	0.8972871	1.354480	-0.2645029
Analysis 2	0.0878054	0.8927953	1.354976	-0.2644143
Analysis 3	0.1360230	0.9055801	1.368586	-0.2275312

Table 5: Comparison of stress intensity factors obtained using different XFEM models for test case with a kinked centre crack.

XFEM Model	Crack Tip 1		Crack Tip 2	
	K_I	K_{II}	K_I	K_{II}
Original	2.3304	-0.4854	2.3303	-0.4855
Analysis 2	2.3634	-0.4883	2.3639	-0.4884
Analysis 3	2.3614	-0.4890	2.3654	-0.4872

5 Case Study

5.1 Problem definition

The final task in this contract was to demonstrate applications of the XFEM capability in analyses of spectral fatigue crack propagations through a case study involving a practical problem, and a test case which was previously analyzed by Martec using the conventional fracture elements in VAST was selected for this purpose.

This test problem was originally considered in a joint project of DRDC Atlantic and UK MOD on the safety of cracked ships for use with the Type 23 Frigates [10]. The aim of the project was to develop and test a crack management process which would allow these frigates to operate in a cracked condition, with appropriate monitoring and inspection, if it was determined to be safe to do so. Part of the process requires assessment of a reported crack using numerical tools and the present test problem had been utilized to assess the accuracy of these software tools [10]. The Martec tool set used in the assessment was the Fatigue II module in the TRIDENT/VAST system. This analysis system was based on the spectral hydrodynamic loading approach, and involved use of a global finite element model representing the overall ship structure and a local finite element model representing the area of interest [11]. The conventional fracture element was employed in this early study [10].

5.2 The global and local finite element models

The Type 23 Frigate global finite element model is shown in Figure 23. This global finite element model was used to compute the global displacements of the ship structure under each of the unit panel pressure loads and inertial forces due to rigid body accelerations. As will be discussed later, these global responses were utilized in a top-down analysis procedure to obtain stress intensity factors for each of the load cases using a local model which included the crack details.

Details relating to the crack considered in the present study are given in Figure 24. In summary, the crack was monitored for a period of 19.04 hours, in which time it grew a total of 12 mm. The initial length of the crack was 85 mm. As noted, a 6-8 sea state with 14 m waves was observed during this period. Using the information available on the crack location, a local finite element model was generated using the top-down analysis option in TRIDENT as indicated in Figure 25 and the crack details were then implemented into the local model (see Figure 26).

In the previous analyses as described in Ref [10], the conventional fracture element in VAST was used and the local mesh around a crack of 88.43mm is depicted in Figure 27 where the crack must coincide with element edges. In the present study, a XFEM model was generated for the same crack as shown in Figure 28 where the crack was allowed to intersect with the element arbitrarily. Because branching cracks are not permitted in the present XFEM element, two of the elements on the coaming near the crack opening were deleted from the model to allow proper definition of the discontinuous displacement fields. We realized that this deletion of elements would cause a change in the local stiffness. However, as these elements were at the end of the crack and almost stress free, this deletion was not expected to have a significant influence on the overall solutions.

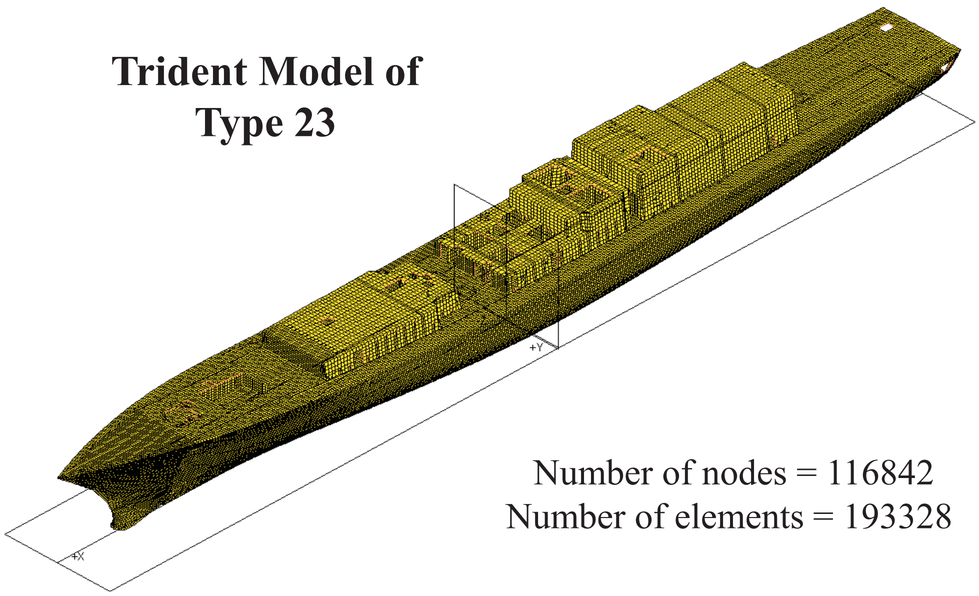


Figure 23: Global finite element model.

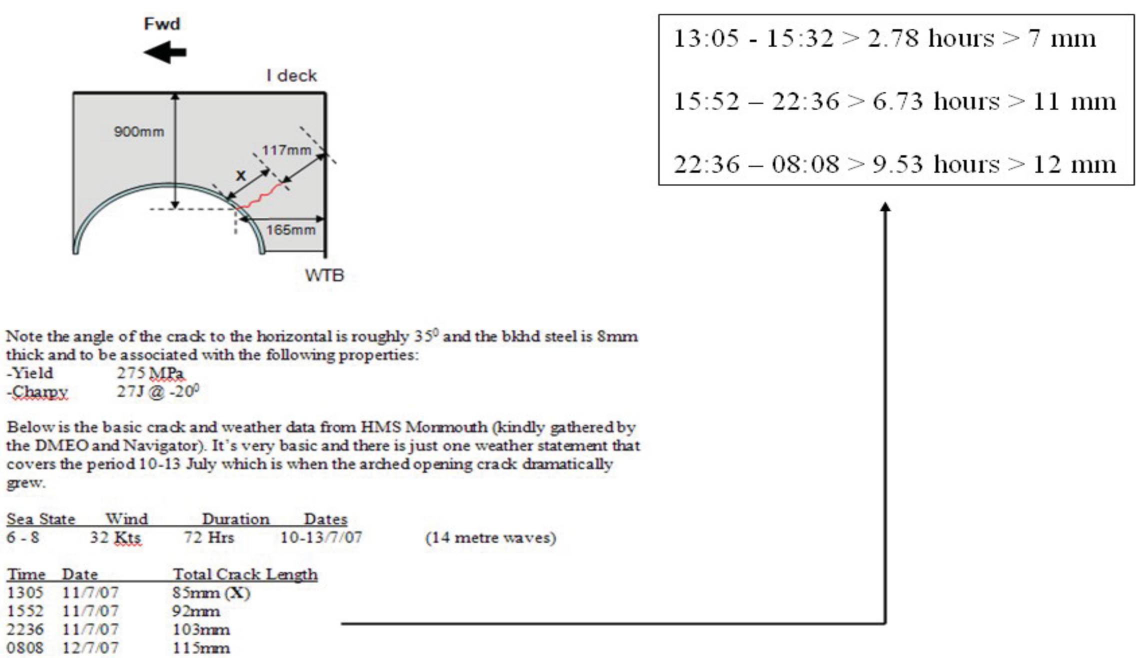


Figure 24: Crack Details.

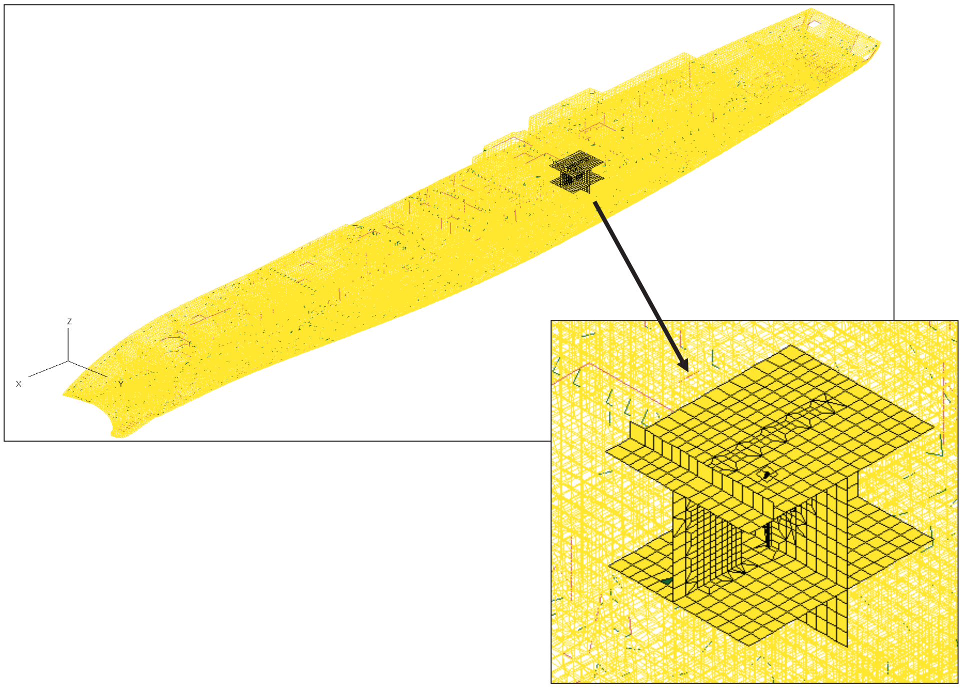


Figure 25: Top-down analysis procedure.

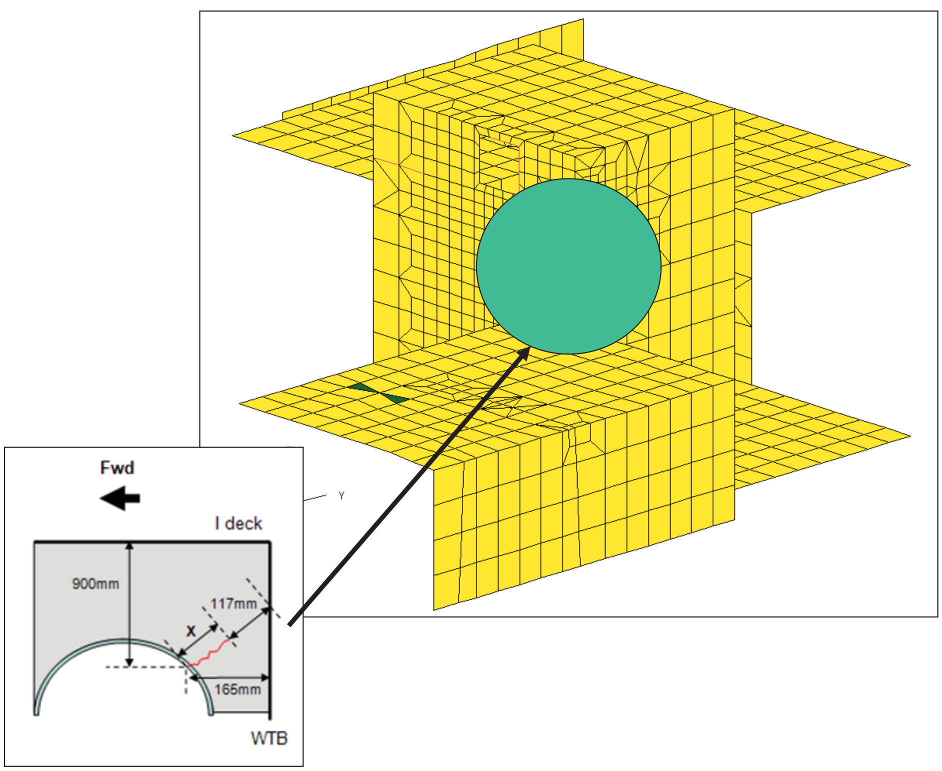


Figure 26: Local finite element model with crack details.

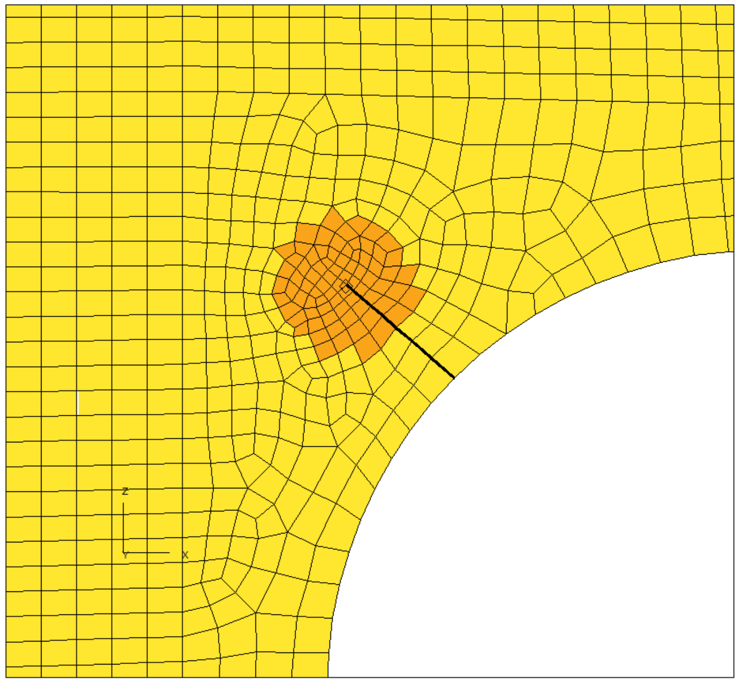


Figure 27: Crack details in local finite element model based on conventional fracture elements.

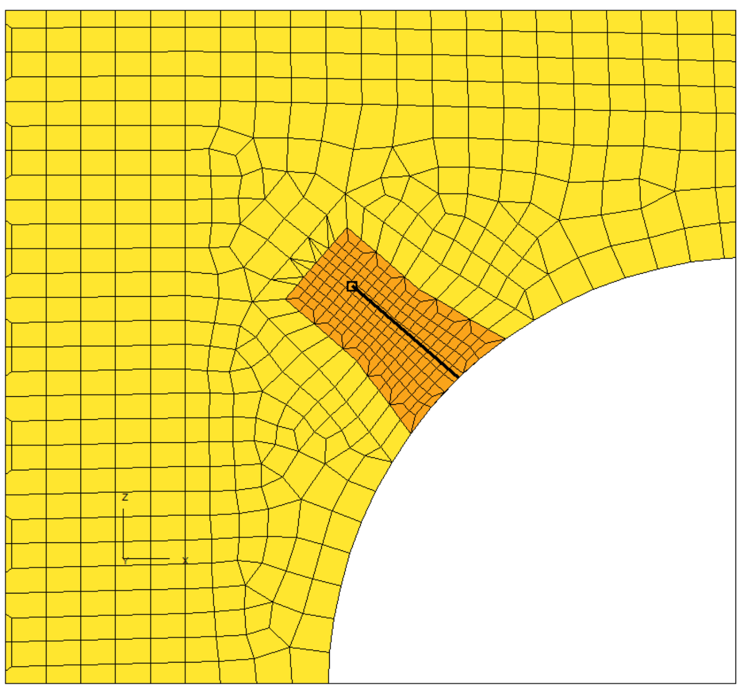


Figure 28: Crack details in local finite element model based on XFEM elements.

5.3 Spectral load analysis

The spectral load analysis was carried out using the hydrodynamics code PRECAL. The input data for PRECAL was generated in TRIDENT and contains a total of 770 facets.

The PRECAL hydrodynamic analysis was performed to obtain pressure and motion response amplitude operators (RAO) for the 3 different sea state cases described in Table 6. In all these cases, the speed and heading were taken as 14 knots and zero degrees, respectively. The sea states were defined by significant wave height and peak modal period according to LR 2005 and are shown in Table 7 [10].

Table 6: Sea state cases considered.

Sea-State Case	Sea State	Probability
1	6, 7 and 8	1/3, 1/3, 1/3
3	7 and 8	1/2, 1/2
6	8 only	1

Table 7: Sea states definitions by significant wave height and peak modal period – LR 2005.

Sea State	Significant Wave Height (m)		Peak Modal Period (sec)	
	Lower Bound	Upper Bound	Lower Bound	Upper Bound
6	4.0	6.0	9.8	16.2
7	6.0	9.0	11.8	18.5
8	9.0	14.0	14.2	18.6

5.4 Calculation of stress intensity factors

A global finite element analysis was performed for 770 unit pressure load cases (one for each hydrodynamic facet) and 6 unit acceleration load cases (3 translational and 3 rotational), for a total of 776 load cases. The nodal displacements from the global analysis were then prescribed to the local models from which the stress intensity factors corresponding to each of the load cases were calculated. The mode I and mode II stress intensity factors predicted by the conventional and XFEM fracture elements are compared in Table 8. These results are found to be in reasonably good agreement. The results from the XFEM elements were consistently lower than those from the conventional fracture elements. This might be a consequence of the relatively coarse finite element models. It should be noted that the signs of K_{II} are different in the results of conventional and XFEM elements in Table 8. This was due to the different definition of the local crack-tip coordinate systems in these elements.

Table 8: Comparison of stress intensity factors (MPa√mm) obtained using conventional and XFEM fracture elements for the first twenty-five unit pressure load cases.

Load Case #	Conventional		XFEM	
	K_I	K_{II}	K_I	K_{II}
1	3.87	-1.28	3.79	1.20
2	7.16	-2.36	7.03	2.22
3	11.09	-3.81	10.90	3.58
4	19.18	-4.49	18.82	4.18
5	21.51	-3.49	21.10	3.20
6	5.30	0.05	5.24	-0.08
7	47.25	-9.66	46.14	8.91
8	51.52	-8.31	50.27	7.58
9	73.77	-13.38	72.02	12.29
10	88.82	-15.60	86.71	14.31
11	125.50	-22.16	122.50	20.33
12	139.20	-24.63	135.90	22.60
13	195.30	-34.70	190.80	31.85
14	213.60	-38.69	208.70	35.54
15	219.30	-39.85	214.30	36.62
16	211.40	-38.82	206.50	35.68
17	244.30	-44.97	238.70	41.35
18	389.30	-71.59	380.40	65.81
19	369.70	-68.13	361.20	62.64
20	422.00	-77.99	412.30	71.71
21	409.20	-76.14	399.90	70.03
22	445.80	-82.85	435.60	76.20
23	501.20	-93.31	489.70	85.82
24	455.00	-85.11	444.60	78.30
25	482.80	-90.52	471.80	83.29

5.5 Crack propagation analysis

The crack propagation analysis was performed using the Fatigue II module in TRIDENT in which a fatigue analysis solver, named Life3D, was utilized based on the following Paris Law equation:

$$\frac{d\,a}{d\,N} = C(\Delta K)^m \tag{12}$$

where a denotes the amount of crack growth, N is the number of cycles, C and m are material constants, and ΔK is the difference between the maximum and minimum values of the stress intensity factor.

In the present crack propagation analysis, the Paris parameters C and m were taken from Ref [12] as $C = 24\text{x}10^{-9}$ and $m = 3$, where da/dN and ΔK are in units of m/cycle and MPa√m, respectively.

In millimetres, the value for *C* was calculated to be 7.59×10^{-13}. These values are known to be suitable for marine environment.

The observed and predicted crack growths over a period of 2.78 hours are compared in Table 9. The numerical predictions presented in the table included solutions obtained for different sea state conditions using stress intensity factors generated by conventional and XFEM fracture elements. As expected, the XFEM predictions are slightly lower than those from the conventional fracture elements due to the consistently lower stress intensity factors as shown in Table 8. However, both methods resulted in reasonable agreement with the observed value.

Table 9: Observed and predicted crack growth over 2.78 hours under different sea states.

Sea State Cases	Crack Growth (mm)		
	Observation	VAST Prediction	
		Conventional	XFEM
6,7,8	7.00	3.25	3.03
7,8	7.00	4.26	3.97
8	7.00	5.99	5.59

6 Conclusions

In this report, some recent extensions of the previously developed 2D XFEM fracture element in VAST have been described in detail. These include removal of some of the limitations in the previous version of the XFEM element, such as the requirement that cracked plates have to be planar and reside in the global X-Y plane and that the crack cannot pass element corners. In order to allow arbitrary orientations, the previous XFEM implementation was reviewed to ensure that all the operations, such as differentiations of the enrichment field, were performed with respect to the element local coordinate system and the internal constraint conditions applied to the enriched nodes were generalized to constrain displacements along the element normal direction. To treat situations where a crack passes through element corners, the nodal values of the enrichment functions were adjusted. These modifications significantly improved the applicability of the XFEM element to deal with practical problems where cracked structural members were often slightly curved and arbitrarily oriented in the 3D space.

In order to improve the usability of the XFEM capability, pre- and post-processing capabilities for XFEM have been developed in the TRIDENT system. In this development, the conventional fracture element (IEC=18) and the XFEM fracture element (IEC=68) were treated in a unified GUI system, which allows definitions of fracture elements and the crack geometry. For the conventional element, crack lines must coincide with element edges and the nodes along the crack must be split. However, for XFEM, the crack geometry can be arbitrarily defined through the use of either pre-defined pseudo-nodes or primitives. The post-processing capability permitted display of average or maximum stresses in elements. For conventional fracture element, stress intensity factors can also be displaced. However, at the present time, the display of stress intensity factors for XFEM element was not yet available. These values can be retrieved from the LPT file. All options in the newly developed XFEM pre- and post-processing capabilities in TRIDENT were verified using numerical examples.

To demonstrate the suitability of the 2D XFEM fracture element for solving practical problems, a case study was performed using a spectral fatigue crack propagation analysis. The test case selected for this case study was a real crack propagation problem on Type 23 frigate that was utilized to evaluate numerical tools for crack propagation simulations in a joint DRDC-UK MoD study on safety of cracked ships. The fatigue analysis was carried out using the unit panel method where stress intensity factors must be evaluated for all the unit panel pressure load cases and the inertial loads resulted from the six rigid body accelerations. In the present study, these stress intensity factors were computed using both the conventional and XFEM fracture elements. Both sets of stress intensity factors were found to result in similar predictions on crack growth and the predictions were in reasonable agreement with the actual measurement of the crack. Given the large number of uncertainties involved in this analysis, we can conclude that the XFEM is indeed a valuable tool for crack propagation analysis.

The case study also indicated that the current XFEM element is unable to treat branching cracks involving intersecting plate members. Due to this limitation, approximations had to be made in the XFEM model. It is recommended that this limitation be removed in future development of the XFEM capability.

References

[1] Jiang, L., *Implementation of 2D XFEM in VAST*, Martec Technical Report TR-10-10, Martec Limited, Halifax, 2010.

[2] Jiang, L., *Improvement of 2D XFEM in VAST*, Martec Technical Report TR-11-48, Martec Limited, Halifax, 2011.

[3] Moes, N., Dolbow, J. and Belytschko, T. "A Finite Element Method for Crack Growth without Remeshing", *International Journal for Numerical Methods in Engineering*, Vol. 46, pp. 131-150, 1999.

[4] *VAST User's Manual*, Version 91, Martec Limited, Halifax, July, 2012.

[5] *TRIDENT User's Manual*, Martec Limited, Halifax, August 2010.

[6] Jiang, L. and Chernuka, M.W. *Further Improvement of the 5-Noded Enriched Fracture Element in VAST*, Martec Technical Note TN-03-04, Martec Limited, Halifax, 2003.

[7] Gifford, Jr. L.N. and Hilton, P.D. "Stress Intensity Factors by Enriched Finite Elements", *Engineering Fracture Mechanics*, Vol. 10, pp. 485-496, 1978.

[8] Tada, H., Paris, P.C. and Irwin, G.R. *The Stress Analysis of Cracks Handbook*, ASME, New York, 2000.

[9] Jiang, L. *Further Enhancement and Verification of the Fracture Mechanics Capability in DSA-VAST*, Martec Technical Note TN-00-07, Martec Limited, Halifax, 2000.

[10] Norwood, M. and Crocker, J. *Testing of a Fatigue Crack Management Process - Type 23 Frigate*, Martec Technical Report TR-09-61, November 2009.

[11] Stredulinsky, D.C. *Proposed Method for Calculation of Spectral Response to Random Wave Loading Using Top-Down Finite Element Modelling*. DREA Technical Memorandum, TM/97/209, November 1996.

[12] Kent, J.S. *Safety of Cracks in Ships – Summary of Methodology*. QINETIQ/D&TS/SEA/TR0702167/1.0, February 2007.

[13] Koks, D. *Explorations in Mathematical Physics*, Springer Science+Business Media,LLC. ISBN 0-387-30943-8. 2006. Ch.4, pps 147 et seq. A Roundabout Route to Geometric Algebra.

This page intentionally left blank.

DOCUMENT CONTROL DATA

(Security classification of title, body of abstract and indexing annotation must be entered when the overall document is classified)

1. ORIGINATOR (The name and address of the organization preparing the document. Organizations for whom the document was prepared, e.g. Centre sponsoring a contractor's report, or tasking agency, are entered in section 8.) Martec Limited 400-1800 Brunswick Street Halifax, Nova Scotia B3J 3J8 Canada	2. SECURITY CLASSIFICATION (Overall security classification of the document including special warning terms if applicable.) UNCLASSIFIED (NON-CONTROLLED GOODS) DMC A REVIEW: GCEC APRIL 2011

3. TITLE (The complete document title as indicated on the title page. Its classification should be indicated by the appropriate abbreviation (S, C or U) in parentheses after the title.)

Further Development of 2D XFEM in VAST

4. AUTHORS (last name, followed by initials – ranks, titles, etc. not to be used)

L. Jiang, M. Norwood

5. DATE OF PUBLICATION (Month and year of publication of document.) July, 2013	6a. NO. OF PAGES (Total containing information, including Annexes, Appendices, etc.) 48	6b. NO. OF REFS (Total cited in document.) 12

7. DESCRIPTIVE NOTES (The category of the document, e.g. technical report, technical note or memorandum. If appropriate, enter the type of report, e.g. interim, progress, summary, annual or final. Give the inclusive dates when a specific reporting period is covered.)

Contract Report

8. SPONSORING ACTIVITY (The name of the department project office or laboratory sponsoring the research and development – include address.)

Defence Research and Development Canada
9 Grove Street
P.O. Box 1012
Dartmouth, Nova Scotia B2Y 3Z7

9a. PROJECT OR GRANT NO. (If appropriate, the applicable research and development project or grant number under which the document was written. Please specify whether project or grant.) 11ge04	9b. CONTRACT NO. (If appropriate, the applicable number under which the document was written.) W7707-125422/001/HAL CU01
10a. ORIGINATOR'S DOCUMENT NUMBER (The official document number by which the document is identified by the originating activity. This number must be unique to this document.) Martec TR-13-30	10b. OTHER DOCUMENT NO(s). (Any other numbers which may be assigned this document either by the originator or by the sponsor.) DRDC-RDDC-2014-C88

11. DOCUMENT AVAILABILITY (Any limitations on further dissemination of the document, other than those imposed by security classification.)

12. DOCUMENT ANNOUNCEMENT (Any limitation to the bibliographic announcement of this document. This will normally correspond to the Document Availability (11). However, where further distribution (beyond the audience specified in (11) is possible, a wider announcement audience may be selected.))

13. ABSTRACT (A brief and factual summary of the document. It may also appear elsewhere in the body of the document itself. It is highly desirable that the abstract of classified documents be unclassified. Each paragraph of the abstract shall begin with an indication of the security classification of the information in the paragraph (unless the document itself is unclassified) represented as (S), (C), (R), or (U). It is not necessary to include here abstracts in both official languages unless the text is bilingual.)

This report is concerned with further development and validation of the XFEM fracture mechanics analysis capability in VAST. In the present work, some of the limitations in the previous version of the XFEM fracture element were removed. The improved XFEM capability permits curved shell geometry with arbitrary orientations and cracks that pass through element corners. Pre- and post-processing capabilities for the XFEM element were developed in the TRIDENT system where the conventional and XFEM fracture elements are treated under a unified framework. In order to demonstrate the suitability of the XFEM element for spectral fatigue crack propagation analyses, a case study was performed using a practical problem involving crack growth in a frigate under certain operation conditions. The fatigue analysis was performed using the unit panel method where the stress intensity factors must be provided for each of the unit panel and rigid-body acceleration load cases. The crack increments predicted by the stress intensity factors from XFEM and conventional fracture elements are in good agreement and they both agree well with the measured data.

14. KEYWORDS, DESCRIPTORS or IDENTIFIERS (Technically meaningful terms or short phrases that characterize a document and could be helpful in cataloguing the document. They should be selected so that no security classification is required. Identifiers, such as equipment model designation, trade name, military project code name, geographic location may also be included. If possible keywords should be selected from a published thesaurus, e.g. Thesaurus of Engineering and Scientific Terms (TEST) and that thesaurus identified. If it is not possible to select indexing terms which are Unclassified, the classification of each should be indicated as with the title.)

XFEM, Fracture Mechanics, TRIDENT, VAST, Spectral Fatigue, Crack Propagation